000475

# VOLUME I
# THE
# PADS™-2000
# USER SELF-TRAINING MANUAL

CAD SOFTWARE, INC.
119 Russell Street
Suite #6
Littleton, MA 01460

(508)486-9521

Rev 1 10/01/90
Rev 0 06/15/90

CAD Software, Inc.© Copyright (C) 1986, 1987, 1988, 1989, 1990 All rights reserved.

This program is the property of and embodies the proprietary trade secrets of CAD Software, Inc. Unauthorized use and copyting are prohibited by law, including United States and Foreign copyright law.

CREDITS:

*IBM is a registered trademark of IBM Corporation.
FutureNet is a registered trademark of FutureNet Corporation.
Redac, Mini, Maxi and Red Boards are registered trademarks of Racal-Recal, Inc., Westford, Massachusetts.
Schema is a registered trademark of Omation, Incorporated.
Orcad is a registered trademark of ORCAD Systems Corporation

# TABLE OF CONTENTS

**CHAPTER 1**
**GETTING STARTED** ..... 1
OUR SELF-TRAINING APPROACH ..... 1
DESIGNER INTERFACE ..... 1
USER MANUALS ..... 2
SYSTEM OVERVIEW ..... 3
ENTERING DATA INTO THE SYSTEM ..... 3
SETUP ..... 4
IN/OUT ..... 4
PLACEMENT ..... 4
CREATE ..... 5
ROUTE ..... 5
CHECK ..... 5
ECO ..... 5
REPORTS ..... 5
CAM ..... 6
ASCII OUT ..... 6
TRAINING FILES ..... 6
LET'S START THE TRAINING COURSE!! ..... 7
WARRANTY AND SUPPORT ..... 7

**CHAPTER 2**
**TUTORIAL 1 - INPUT/OUTPUT AND SETUP** ..... 9
2.1 INTRODUCTION ..... 9
AND NOW LET'S GET STARTED IN THE USE OF THE PADS-2000 SOFTWARE!! ..... 10
2.2 CRT PRESENTATION ..... 14
WORKING AREA ..... 14
2.3 INPUT/OUTPUT ..... 19
2.4 THE SETUP ..... 21
2.4.1 DISPLAY ..... 22
2.4.2 PADS ..... 24
2.4.3 SIZES ..... 24
2.4.4 PARAM(ETERS) ..... 25
SHOW CURRENT LEVEL ..... 25
CURRENT NET HIGHLIGHT ..... 25
TEAR DROP ..... 26
COPPER POUR ..... 26
DOT GRID ..... 26

REAL WIDTH....26
THERMAL LINE WIDTH....26
COPPER HATCH GRID....27
COPPER POUR CLEARANCE....27
CLEARANCE....27
BACKUP INTERVAL....27
MAX ROUTING LEVEL....28
2.4.5 NET ATTR(IBUTE)....29
2.4.6 MACROS....30
2.4.7 SR PARMS....30
2.4.8 UNITS....31
2.4.9 ORIGIN....32
2.4.10 SCREEN MANIPULATION....33
2.4.11 MODELESS COMMANDS....35
2.5 JOB OUT....35

**CHAPTER 3**
**TUTORIAL 2 - CREATING ITEMS AND PARTS PLACEMENT**....49
3.1 THE CREATE ROUTINE....49
BOARD OUTLINE....49
TEXT....51
CREATING 2-D LINES....52
3.2 CREATING A BOARD DATABASE....54
SETUP....55
3.3 PRACTICING THE INTERACTIVE PLACE....58
MOVE....59
MATRIX MOVE....60
ROTATE....61
OPPOSITE....61
LENGTH MIN ON THE FLY....62
ALTERNATE....63
GLUE....63
MOVE NAME....64
3.4. PRACTICING THE AUTO PLACE....64
3.5 PARTS PLACEMENT ON THE TRAINING BOARD....67
CONNECTOR PLACEMENT....68
MOIRE PAD....69
UNFIXED (MOVEABLE) COMPONENTS....70
IC PLACEMENT....73
AUTOPLACEMENT....74
DISCRETE PARTS PLACEMENT....75

DECOUPLING CAPS ..... 75
3.6 GATE AND PIN SWAP ..... 76
3.7 GOODNESS OF PLACEMENT EVALUATORS ..... 78
3.8 RENAMING OF PARTS ..... 78
3.9 GROUP OPERATIONS ..... 80
3.10 GOOD COMPONENT PLACEMENT ..... 81
3.11 ADDITIONAL AUTO-PLACEMENT ..... 83

**CHAPTER 4**
**TUTORIAL 3 - CONNECTION ROUTING**
4.1 THE ROUTE ROUTINE ..... 89
4.2 MODIFY TRACK ..... 97
4.3 T-ROUTING ..... 98
4.4 THE SHOVE ROUTER ..... 101
4.5 ROUTING THE TRAINING BOARD ..... 103
4.5.1 PLACEMENT/ROUTING GRID SYNC TEST ..... 103
4.5.2 POWER AND GROUND ROUTING ..... 104
4.5.3 SIGNAL HEURISTIC ROUTER ..... 108
4.5.4 SIGNAL MAZE ROUTER ..... 109
4.5.5 VIA MINIMIZATION ..... 110
4.5.6 ROUTING GRID SHIFT ..... 111

**CHAPTER 5**
**TUTORIAL 4 - REPORTS, CHECKING AND CHANGE ORDERS** ..... 117
5.1 REPORTS ..... 117
5.2 CHECKING ..... 117
5.3 ENGINEERING CHANGE ORDERS ..... 120

**CHAPTER 6**
**TUTORIAL 5 - COMPUTER MANUFACTURING AIDS** ..... 131
6.1 ARTWORK, LAYER 1 ..... 132
6.2. ARTWORK, LAYER 2 ..... 134
6.3 ASSEMBLY DRAWING ..... 134
6.4 SOLDER MASK ..... 134
6.5 DRILL DRAWING ..... 134
6.6 PHOTOPLOTTER ARTWORK ..... 135

CHAPTER 7
TUTORIAL 6 - ASCII FILES, THE PARTS LIBRARY AND DESIGN-ON-THE-FLY ..... 139
7.1 ASCII INPUT ..... 139
7.1.1 PARTS LIST ..... 139
7.1.2 CONNECTION LIST ..... 141
7.1.3 READING ASCII IN ..... 141
7.2 ASCII IN - ASCII OUT ..... 142
7.3 DESIGNING-ON-THE-FLY ..... 144
7.3.1 ADD PARTS AND CONNECTIONS ..... 145
7.4 LIBRARY PARTS ..... 146
7.4.1 PART TYPES (Using PLIB) ..... 148
7.4.2 Part Type (Using PADS-2000 CREATE) ..... 151
7.4.2.1 Creation of a Multi-Gated (74LS02 IC) Component ..... 152
7.4.2.2 Creation of a Discrete Part ..... 156
7.4.3 Modify Part Type ..... 161
7.4.4 PART DECALS ..... 161

CHAPTER 8
TUTORIAL 7 - MULTI-LAYER DESIGN PROCEDURES ..... 167
8.1 MULTI-LAYER SETUP ..... 167
PARTS PLACEMENT ..... 172
CONNECTIONS ROUTING ..... 172
CHECKING ..... 173
ALTERNATE METHOD ..... 173
CAM ..... 174
8.2 MULTI-LAYERS (IN EXCESS OF FOUR LAYER) ..... 175
8.3 MULTI-LAYERS (BURIED/BLIND VIAS) ..... 178
8.4 TRACKS ON PLANES ..... 180

CHAPTER 9
TUTORIAL 8 - SMD DESIGN ..... 183
9.1 INTRODUCTION ..... 183
9.2 EXAMPLE ONE - 4-LAYER, MULTI-LAYER ..... 183
PLACEMENT OF PARTS ..... 184
BOTTOM LAYER ..... 184
TOP LAYER ..... 184
AUTO PLACE OF IC's ..... 184
INTERACTIVE PLACE OF IC's ..... 185
SETUP ..... 186
PAD STACKS ..... 187
POWER AND GROUND PLANES ..... 187

CREATING THE POWER/GROUND PLANES ..... 190
POWER PLANE ..... 191
MEMORY CONNECTIONS ROUTING ..... 191
ROUTING THE REMAINING SIGNAL CONNECTIONS ..... 194
CHECK ..... 195
CAM ..... 196
9.3 MULTI-LAYER BOARD, EXAMPLE #2 ..... 196
SETUP ..... 196
POWER AND GROUND PLANES ..... 198
MEMORY ARRAY CONNECTION ROUTING ..... 198
NON-MEMORY CONNECTION ROUTING ..... 199
PAIRED LAYERS 5 AND 6 ..... 199
ALL ROUTING LAYERS ..... 200
CHECKING ..... 200
CAM ..... 201
CONCLUSION ..... 201

**CHAPTER 10**
**TUTORIAL 9 - JOB FILES AND ARCHIVAL MAINTENANCE** ..... 205
DATABASE FILES ..... 205
FILE DELETION ..... 206
ARCHIVAL FILE STORAGE ..... 206
SYSTEM LIBRARY FILES ..... 207
PART TYPE COPY ..... 208
PART DECAL COPY ..... 208
LINE ITEMS TRANSFER ..... 209
BACKUP FILES ..... 209

**CHAPTER 11**
**TUTORIAL 10 - THE MACRO ROUTINE** ..... 211
11.1 INTRODUCTION ..... 211
11.2 MACRO COMMANDS ..... 211
11.3 TYPES OF MACROS ..... 212
11.4 CREATING MACROS ..... 212
11.5 CREATING AN AUTO MACRO ..... 213
11.6 USER MACROS ..... 214
11.7 MACRO LANGUAGE ..... 215
11.8 LOADING USER MACROS ..... 216
11.9 USER DESIGN MACROS ..... 217
11.10 HOW TO REPLAY A USER MACRO ..... 218
11.11 WHAT ARE MENU MACROS ..... 218

INDEX

APPENDIX 1 - POSSIBLE ERRORS AND CAUSES

# CHAPTER 1 - GETTING STARTED

## INTRODUCTION

*Congratulations!! You have selected the PADS-2000 Program, the most productive Printed Circuit Board Design software packages available--at any price! Now that you have your Program, your next task is to learn how to use it effectively--and to do this in the least possible time.*

## OUR SELF-TRAINING APPROACH

We at CAD Software, Inc. have made a major effort in developing a self-training approach that will enable you to learn to use the system without a great expenditure of time and effort. The approach we use is an integrated, dual thrust effort involving (1) a highly efficient Designer Interface at the screen, and (2) special attention to the organization and content of the User Manuals.

The important thing is that you read and study these manuals before rushing into trying to use the system. Experience with thousands of users of PADS-2000 has shown that the single most common problem is with the user that attempts to use this sophisticated system without adequate study. The User Manuals have been prepared for your use! Use them, please!

## DESIGNER INTERFACE

The Designer Interface with the CRT screen is self-coaching and easy to use. The system commands are self evident, familiar and logical to a Printed Circuit Board designer. Example: PART MOVE, ROTATE, GLUE, etc. are all sub-commands within the PLACE routine--a Board Designer will be able to select and use these commands without detailed instructions in their use.

To simplify and streamline the Designer Interface, the command structure has been organized to duplicate the designer's thought process

during the design, i.e., Parts Placement, Routing, Checking, etc. All commands associated with each routine have been grouped together, so that at any one time, only those commands pertinent to the operation, are presented to the designer for his decision and action, as necessary.

A unique feature to the PADS-2000 Designer Interface is a "Prompt Window" at the bottom of the screen which continually coaches the designer as to the information required for the particular action, including the format of the designer's response.

## USER MANUALS

The second thrust in our effort to ease the learning process is the User Manuals, their content and organization. There are two User Manuals: (1) the Self-Training Manual and (2) the Reference Manual.

1. Volume I, the User Self-Training Manual has been prepared as a step-by-step, do-it-yourself instructional manual, which used with the Software and computer (and a specially designed Training board included with the software), leads the designer through the different steps in the complete design of a Board. This Training board has been prepared so as to include virtually every situation a designer would encounter during design of a representative board. As a result, upon completion of the Practice board, you will have a good appreciation for the capability of the PADS-2000 Software in virtually any situation encountered during design.

The philosophy of the User Self-Training Manual is that you will learn the system better and more rapidly by the actual design of a board using the Self-Training Manual, the training board and your own computer, than you will by only reading about how to use the system. Basic tutorials using the system in the design of a two-layer board are followed by Advanced tutorials covering multi-layer board design, System Library and Archival Maintenance in the Self-Training Manual.

We strongly urge you to self-train yourself in the use of PADS-2000 by following the tutorials in Volume One.

2. Volume II, the User Reference Manual is an all-inclusive, detailed document containing "all the facts" on the software program. It is to be "referred to" for detailed instructions in those instances where the

designer wishes detailed knowledge of a feature beyond that provided by the Self-Training Manual.

## SYSTEM OVERVIEW

We believe you will better appreciate the step-by-step contents in the Self-Training Manual if, before you start your training, if you are provided with an overview of the PADS-2000 software.

The following discusses the relationship of the various Routines of the PADS-2000 software, together with an approximate sequence of use of these Routines, during the design of a board.

## ENTERING DATA INTO THE SYSTEM

There are a number of sources that can be used to enter data into the system to start a board, all but one require the generation of a Net list and a Parts list. One source of the Net list and Parts list is from the FutureNet™ Logic Capture system or systems such as Orcad™ and Schema™ that output FutureNet format; another source is from other Logic Capture systems that output a Parts list and Net list in ASCII format. A third source is the PADS-LOGIC Logic Capture Program (used in this Training Manual). A fourth Input source is from a manually created schematic from which the Net list and Parts list were created and inputted using the PADS-2000 Program Editor (PEDIT). A fifth input method is to enter the Parts and Connection Lists directly at the CRT using the Design-On-the-Fly Routine.

In the training exercise, the Parts list and Net list created in PADS-LOGIC, are inputted to the system through the ASCII IN subroutine. As the lists are read in through the ASCII IN subroutine, a number of things occur automatically:

First, a check occurs on the input data and any errors found are printed and must be corrected before the data is accepted. You will derive great comfort from this early checking of the data since early correcting of errors is much more efficient than finding and correcting them at some later time in the design sequence.

Second, a "Setup" file is created and integrated with the board database and stays with the database throughout the design effort. This Startup file contains Default values such as Text, Line and Grid size, Number of layers, etc.

Third, the Input Parts list is used to extract the data associated with Part Type and Part Decal from the System Parts Library and create the board database.

Finally, the board database, created by the Parts and Connection List, is displayed at the screen for your inspection and use.

## SETUP

The first step in the design sequence is the adjustment of the standard System SetUp as required for the requirements of the particular board under design, i.e., grid, etc., using the SETUP routine (we do this in Tutorial 1).

## IN/OUT

During the various stages of design, you will want to store or save interim files of your design on the hard disk. The IN/OUT routine is used to write files to disk, as well as to recall files from disk (Tutorial 1).

## PLACEMENT

Next, the Parts are placed on a board in the most desirable location using the Interactive and Automatic features of the PLACE routine. Placement evaluators are used to decide the best placement. Parts can be moved, rotated, glued down and swapped to the opposite side of the board. Automatic routines swap parts to different locations under Designer direction (Tutorial 2).

## CREATE

The routine of CREATE is used to create various items on the board such as Text, 2-D Lines, Copper areas, Part Decals, etc (Tutorial 2).

## ROUTE

After the components are placed, the rubber band connections are converted into Tracks on different layers of the board using both Automatic and Interactive Routing. Up to 30 layers are possible using different Line Widths as appropriate. Both Thru vias, Buried vias and Blind vias are supported. Additionally, Standard size vias and Micro size vias can be used (Tutorial 3).

## CHECK

Checking is used to determine any violations of the minimum Air Gap spacing between various items on the board. Correction of spacing violations can be made using Modify Route of the ROUTE routine (Tutorial 4). A comparison of the original Connection and Parts Lists can be made against the current data base.

## ECO

Engineering Change Orders consisting of the addition or deletion of components, connections, changing of Parts, Renaming of Parts, etc., is accomplished using the ECO routine (Tutorial 4).

## REPORTS

At any stage during the design of a board, a series of statistical and status reports can be generated and used for analysis of the board (Tutorial 4).

## CAM

The various outputs of the system are generated utilizing the CAM or Computer-Aided-Manufacturing routine. Among the outputs of the system are Artwork, Ground and Power Planes, Silkscreen, Pads Only, Soldermask, Assembly Drawing, Drilling Drawing, etc. (Tutorial 5).

Four forms of outputs are possible:

1. Plots of the above on the matrix printer associated with your system (these generally are used for check plots),

2. Plots of the above on Laser Printer.

3. Scaled Wet ink artwork plots can be generated on a wet ink plotter and photo reduced to 1:1, thus eliminating the need for a photoplotter,

4. A Gerber file on floppy disk that can be used to drive a PC equipped, Gerber photoplotter or Laser photoplotter.

5. An Excellon Drill file on floppy disk that can be used to drill the board.

## ASCII OUT

The database representing the board can be interfaced to other systems using the ASCII OUT routine which converts the database into an ASCII format files.

## TRAINING FILES

The following files used during your training have been included in the subdirectory \FILES:

| | |
|---|---|
| TPART.ASC | Parts in Training Board in ASCII |
| TCONN.ASC | Connections in Training Board in ASCII |
| TPLACE1.JOB | Practice Placement Board |
| TAPLACE1.JOB | Automatic Placement Board |
| TROUTE.JOB | Practice Route Job |
| TCHECK.JOB | Practice Check Job |

| | |
|---|---|
| TONTHEFL.JOB | Practice On-the-Fly Job |
| TGROUP.JOB | Practice Group Job |
| TPRACTIC.JOB | Setup Practice Board |
| SMDBO.JOB | Board Outline for SMD |
| SMDPARTS.ASC | SMD Parts List |
| SMDCONN.ASC | SMD Connections |
| PCB.MAC | Macro Files |

## LET'S START THE TRAINING COURSE!!

At this time, you should have your computer loaded with the PADS-2000 software, the computer should be operational, you should have Volume 2, the User Reference Manual, available for reference, and at the end of this chapter, turn to Tutorial 1, Chapter 2 of Volume 1, the User Self-Training Manual.

Each of the self-training Tutorials are structured for a particular routine or routines. For example, Tutorial 1 addresses what you see at the CRT screen, the IN/OUT routine, and the SETUP routine.

We suggest that generally you spend two to three hours on each Tutorial, insuring that you understand the contents of the Tutorial before moving to the next Tutorial.

Where the Tutorial calls for practice of a particular item, be sure to do the Practice. Use Volume II, the Reference Manual, if you want to explore a topic in more detail. Again, USE THE MANUALS TO LEARN TO USE THE SYSTEM. DON'T, REPEAT, DON'T TRY TO LEARN TO USE THE SYSTEM OVER THE PHONE WITH OUR SUPPORT DEPARTMENT, PLEASE!

## WARRANTY AND SUPPORT

At the end of this chapter is a blue and pink card. The Blue card is your 30-day Warranty Registration card. As a part of your software purchase, you are entitled to assistance during the initial loading of your software and for 30 days subsequent to the purchase while you are using the software. This is available only if you send in the blue 30-day Warranty Registration card. NOW, while you are thinking about it, we

suggest you mail back the blue 30-day Warranty Registration card to CAD Software, Inc.

*The Pink card describes an Extended Support Contract which CAD Software offers you, to go into effect upon the completion of your 30-day Warranty Registration period. This Extended Support Contract is for the following 12 months and provides unlimited telephone consulting services with our Hot-line Support Group for any problems or questions you may have. You will receive no-cost updates of all enhancements or improvements as well as bug fixes that are released during the next year. As a suggestion, before your 30-day warranty period is over, give considerable thought to the purchase of the Extended Support Contract. You will find it most worthwhile.*

000475

Your Registration Serial Number

# PADS-2000 EXTENDED SUPPORT CONTRACT

As a user of **PADS-2000** you may, upon completion of the 30-day warranty registration, purchase an Extended Support contract for the following 12 months. The cost of this contract is $750.00 per year (U.S. and Canada) and the benefits far outweigh this small cost. The income from this Extended Support is used for:

(1) Maintaining a staff of specialists to respond to any questions you may have, and

(2) NO-COST enhancements to your **PADS-2000** Software.

A **PADS-2000** Extended Support Contract entitles you to the following:

(1) Unlimited telephone consulting on any problems or questions you may have relative to potential software bugs, special work-arounds you may want to use for unique or new applications, etc.

(2) Written correspondence concerning any questions or problems you have regarding the use of the software.

(3) Free updates of the **PADS-2000** software you have purchased. One or more updates per year of the software are issued, incorporating enhancements or improvements as well as fixes for bugs reported by yourself and other users throughout the world.

The **PADS-2000** Extended Support Contract is applicable to **PADS-2000**. It is not applicable to the PADS-LOGIC or the PADS-SUPERROUTER software.

How to Order: In order to maintain uninterrupted continuity with your 30-day Support Agreement, use the following procedure:

(1) Complete the Extended Support Contract today and mail to CAD Software, Inc., 119 Russell Street, Suite #6, Littleton, MA 01460 with your check for $750.00 (or use Visa/ MasterCard or American Express).

(2) CAD Software, Inc. will start your Extended Software Contract and enter your account into our Support computer.

(3) At any time you need to contact us, call our Hot Line (508) 486-3328 and provide us with your Registration Serial Number. We will assist you with whatever questions you may have. Before calling CAD Software, however, if you have purchased your software and support from a CAD Software, Incorporated authorized Dealer, you should seek help from your local dealer. If your local dealer is not in a position to resolve your problem, please contact us at (508) 486-3328.

(4) We maintain a modem over which you can send us Files you wish us to investigate. The Modem Number is (508)486-9549. Call before sending a file.

# PADS-2000
# EXTENDED SUPPORT CONTRACT

MAIL TO: CAD Software, Inc.
119 Russell Street
Suite #6
Littleton, MA 01460

or FAX TO:
(508)486-8217

_____
Your Registration Serial Number

1. Enroll me in an Extended Support Contract for my **PADS-2000** software. Payment of $750.00 is enclosed___. Charge my American Express/Visa/Master Charge Account No._____, Expiration date_____.

2. My hardware is:

| | Vendor | Model |
|---|---|---|
| CPU | _____ | _____ |
| Memory | _____ | _____ |
| Mouse | _____ | _____ |
| Printer | _____ | _____ |
| Graphics Card | _____ | _____ |
| Printer | _____ | _____ |

3. My software purchased _____
(date)

RETURN TO:

_____
Name

_____
Company

_____
Address

_____

_____

Telephone No._____Ext.____
Fax No._____

# CHANGE OF ADDRESS

We at CAD Software, publish a quarterly Newsletter, send out new software updates, prepare letters on new products and new releases, etc.

These items are sent directly to the original person to whom the software was sent when purchased.

If that person is promoted out of CAD, or changes department or corporation, this very informative and valuable information sent out by CAD Software may very well become lost. We ask you, therefore, if there is a change in the "responsible" person for the PADS system, or a change in address, to advise us on this form. In this fashion, we can continue to keep you and your firm advised about new programs, updates, etc.

Please complete the following and return to:

CAD Software, Inc.
119 Russell Street, Suite #6
Littleton, MA 01460

---

**PADS-LOGIC** Y/N Serial No._____
**PADS-2000** Y/N Serial No._____

Original Responsible Person:
_____Tele:( )_____

Address_____

_____

New Responsible Person:
_____Tele:( )_____

Address_____

_____

# Chapter 2

# TUTORIAL 1 - INPUT/OUTPUT AND SETUP

## 2.1 INTRODUCTION

(120 - 150 minutes at the screen)
This is the first of a series of several Tutorials aimed at getting you started in the use of the PADS-2000 Software and doing so with a minimum amount of effort on your part. Although the PADS-2000 Software is exceedingly rich in features and could necessitate a considerable expenditure of time on your part in learning to use the software, we believe that the solution we will be using for self-training is most desirable in terms of minimizing your effort while learning to use the software.

We have arranged these Tutorials so that you will be able to cover each Tutorial (except for the Routing routine) in about two-three hours or less of intensive work on your part. Each Tutorial covers one, or more, of the major routines used in the software.

The first Tutorial covers the Screen Presentation and the two Routines of IN/OUT and SETUP. We shall use a 20 IC practice board for all training sessions, and in the process we will go through the entire design process. Although somewhat small in size, we have attempted to incorporate into this practice board virtually every real life situation a designer would experience during the process of board design. Accordingly, by the time you finish the Practice Board, you will be fully capable of using PADS-2000 for "real life" board design.

Following the initial basic Tutorials on two-layer board design are some Advanced Tutorials. These Advanced Tutorials cover ASCII File Input, System Library Maintenance, Design-On-the-Fly, Multi-Layer Design with SMD techniques and File Maintenance.

From time to time you may receive Error Messages for various reasons. Refer to Appendix 1, this volume, or Appendix A7 of Volume II for an explanation of these messages.

AND NOW LET'S GET STARTED IN THE USE OF THE PADS-2000 SOFTWARE!!

We assume that your hardware is installed and powered up, and the PADS-2000 Software has been loaded on to your hard disk. Initialize the PADS-2000 software by typing in the following in response to the C:> or C:\PADS> prompt:

**2000<CR>**

You may receive the following message:

> A previous crash of a 386 application has probably left some last clusters on your hard drive. It is therefore suggested you run CHKDSK/F or an equivalent utility which is compatible with your disk manager.

The background on this message is as follows: The Virtual Memory Manager allocates and reserves a portion of your hard disk to the program and the Board Database. Should you (1) have a system crash, or (2) shut down your system (power off) with the program loaded, that portion of the hard disk allocated by Virtual Memory will be left with "lost clusters" on the hard disk. With time, repeated crashes or power off without EXIT will occupy ALL of your hard disk and you will be unable to allocate any of the hard disk.

When you receive this message, in response to the DOS prompt, type:

**CHKDSK/F <CR>**

Most Disk Managers will respond to this prompt; some will NOT, in which case you do not use CHKDSK, but instead use the equivalent Check Disk program for your Disk Manager.

To the prompt:

**CONVERT LOST CHAINS TO FILES: Y/N**

Respond N and your Disk will be cleared of clusters. An "N" deletes the clusters. A "Y" will result in a file being created, which you, in turn, can delete from the DOS directory. Unless you have a good reason for using Y, we suggest you use N.

The PADS-Shell display containing Ports or Windows for PADS-LOGIC, PADS-2000, and PADS-SUPERROUTER is displayed. Using the Mouse, select the PADS-2000 Port.

The following query will appear:

**ENTER MEMORY ALLOCATION FOR PCB DATABASE [1-20 MBYTES]**

Be aware of the following in your memory allocation response.

PADS-2000 uses Virtual Memory. This means that if there is not sufficient RAM memory on your system to load the program and the job which is being designed, software will page out to the hard disk. Virtual Memory will automatically swap out those parts of the program and/or the design which you are not actively using at a given moment. Virtual Memory is efficient and thus you still retain much of the maximum performance of the software even when you have only 50% of the RAM required for your design, although there will be a slow down in performance.

There are three main areas of the software that use Virtual Memory:

a. System Program - This is a fixed overhead that cannot be controlled by the user.

b. PCB Job Database - The amount of memory allocated is user specified upon entering the program.

c. Display List - The program allocates this automatically as required. The size of this memory is dependent on the actual complexity and size of the design.

The sum total of these three main areas is the total amount of Virtual Memory required by the software. If the total amount of memory

required exceeds the amount of RAM available on your system, then the software will start paging to disk.

The following is an approximation of how memory is used in PADS-2000:

| | |
|---|---|
| System Memory | 76K |
| PADS-2000 Program | 1,560K |
| Database | Variable* |
| Display List | Variable** |

* Assume 1 to 10K bytes per IC depending upon number of layers.
**Assume .75 to 1KB per IC for the Display List.

Example: On a 400 IC board, with 4 MB of RAM in your system. You allocate 1.5 MB in response to the database allocation prompt. The use of the allocation is as follows:

| | |
|---|---|
| System | 76K |
| Program | 1,560K |
| Database | 1,536K |
| Display List | 2,096K |
| Total Memory Allocated | 5,268K |
| Total Memory Available | 4,096K |
| Disk Memory Used | 1,172K |

This example would result in light use of the disk, and hence the system response would be nearly optimal.

Respond with the memory you have in your CPU followed by <CR>.

Our Suggestions:

| Board Content | RAM on Board |
|---|---|
| Up to 500 IC Board | 6MB RAM |
| Up to 1000 IC Board | 8MB RAM |
| Up to 1500 IC Board | 12MB RAM |
| Above 2000 IC Board | 16MB RAM |

After entering the Memory Allocation, a copyright notice and data relative to your system will appear along with the prompt:

**PRESS ANY KEY TO CONTINUE**

The PADS-2000 software will now be loaded into memory and after the copyright display, the Screen Presentation will be as shown in Figure 2.1

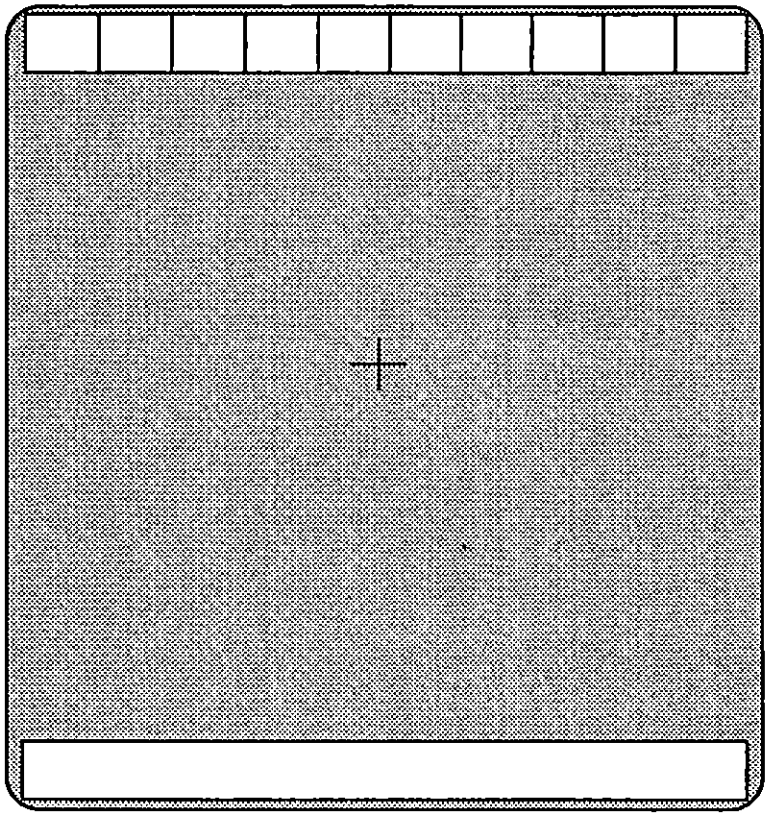


*Figure 2.1 - SCREEN PRESENTATION*

There are three major areas on the screen that you should become familiar with. These are:

1. The Working Area covered by a dot grid spaced at intervals in X and Y, on which you will perform the design.

2. The horizontal row of mini-windows located at the top of the screen containing the Command menu.

3. The Prompt Line located at the bottom on the screen, and used to provide you with messages, information, confirmation requests, etc.

Caution: Error messages are presented on the Prompt Line. Consult Appendix 1 of this Volume and Appendix A7 of Volume II for the meaning of error messages.

## 2.2 CRT PRESENTATION

WORKING AREA

The working area of the system is 56.3" x 56.3" or 143 cm with the Origin in the lower left hand corner of the screen. Cursor movement relative to the Origin is shown in a pop-up window presented by the command CTRL-L. Movement is in Grid units at the grid setting at the time. Move the mouse around and notice the cursor follows the mouse movement and the cursor location is shown in the pop-up location window.

The Menu Window displays either the major options and, their subroutines, as used to control the system during the design activities.

The PADS-2000 Program uses a Hierarchical Command Menu structure. The highest rank in the Hierarchy is the Major Options, of which there are nine different Options, plus EXIT, as follows:

| | | | | |
|---|---|---|---|---|
| **1 In/Out** | **2 Setup** | **3 Create** | **4 Place** | **5 Route** |
| **6 Check** | **7 ECO** | **8 Reports** | **9 CAM** | **10 Exit** |

When any of the nine Major Options is selected, the Major Options are replaced with the next highest rank of subroutines, (the commands associated with the Major Option that was selected).

The Subroutines, in turn, have from one to nine sub-Subroutines associated with each of them. When a Subroutine is selected, the Subroutine menu is replaced with the sub-subroutine associated with the selected subroutine.

EXIT (F10) and Mouse Button Three, in all instances, are used to exit that rank and go to the next higher rank. For example, if in a sub-subroutine, EXIT will move the system from the sub-subroutine to the subroutine containing the sub-subroutine. If in a Subroutine, EXIT will move the system from the sub-Subroutine to the Major Option containing the Subroutine. EXIT (F10), when at the highest or Major Option level, permits the "leaving" of the PADS-2000 Program and returns the system to the PADS-2000 Shell.

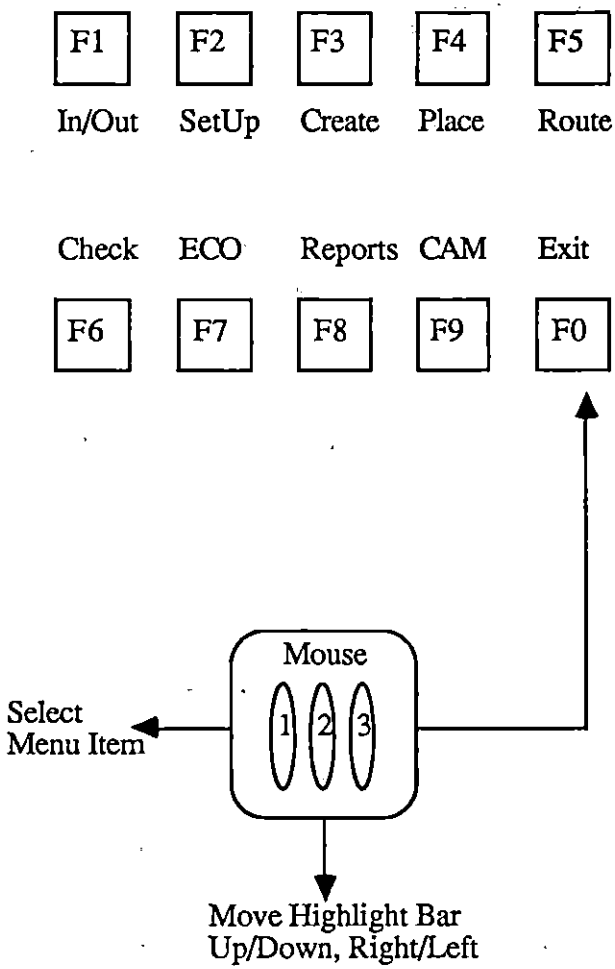



*Figure 2.2 - MENU COMMAND SELECTION*

The Command Menu items are selected with either the "F" Function Keys located on the keyboard, or the mouse buttons, as show in Figure 2.2. Menu routines, subroutines and options, 1 through 10 are selected with the F1 through F10 buttons located on the keyboard. Alternatively, the three-button mouse, is mapped such that in addition to moving the cursor, its buttons can select any of the ten Command Menu choices. The left-handed mouse button (button #1) is mapped to select, or activate the routine, subroutine or option over which the Menu Highlight Bar has been placed. The middle, or mouse button #2 is used to move the Menu Select Highlight Bar over the menu items to be "selected" with mouse button #1 or by "clicks" of the middle mouse button. Finally, the right-hand mouse button, (button #3) is mapped to the EXIT routine in the same fashion as Function Key F10.

Depressing mouse button #2, together with movement of the mouse advances the highlight bar to different Command Menu positions.

A third way of moving the Highlight Menu Bar is by individual "Clicks" of Mouse Button #2. Using this method, a "click" of button #2 will advance the Highlight Bar from Left to Right across the Menu Commands, with each click advancing the Highlight Bar one menu item. To use this technique, you must change your software slightly, as follows:

1. Exit from PADS-2000 and return to the DOS prompt.

2. Be in the \PADS directory. If necessary, enter: CD\PADS.

3. Delete the file called "CURSOR." (Enter: DEL CURSOR).

When you re-enter PADS-2000, each "click" of the middle mouse button will "step" the highlight bar across to the next menu item.

If, after trying this method, you find you wish to return to the original operations, proceed as follows:

1. Exit from PADS-2000 and return to the DOS prompt.

2. Be in the \PADS directory. If necessary, enter: CD\PADS

3. Create a file called "CURSOR," as follows:

Enter: PEDIT (This will bring up PEDIT.) The file will not contain data.

Select WRITE (F10) and use CURSOR for new filename, followed by <CR>. (This will create a file (without data) called CURSOR.)

Select QUIT (F9). (This will exit PEDIT and bring you back to DOS.)

When you re-enter PADS-2000, the mouse will function as in the original mode.

In summary, F buttons F1 through F10 can be used, depending upon the choice, to select the ten Command Menu choices. The mouse, in addition to controlling the cursor, can be used to select Menu Commands. We suggest you use the mouse for this activity.

The designer will find that a position with the left hand over the F buttons, and the right hand on the mouse is the most comfortable operational position; and with practice, the simultaneous selection of the proper routines and the control of the cursor, will become rapid and efficient. Alternately, the designer can use only one hand and the mouse to position the cursor, as well as to select the Menu item. It is suggested that you attempt both methods, and utilize the technique best suited to your personal reaction.

Let's now get some practice with the Command Menu. Select any routine, let's say CREATE, (F3). Depress the middle button and move the mouse so the Red highlight is over CREATE, the select with button one. Note that the nine major options in the window are replaced with the nine sub-subroutines of CREATE. Let's now select one of the sub-subroutines, say 2-D LINE (F4). Note that the CREATE options are replaced with the options used to create a figure by drawing a line. Let's now select BOARD (F2), which provides an option by which we can create a Board Outline. Now select NEW CIRCLE (F2), which provides the ability to create a circular board.

To go back to the major option level after the above effort, select (F10), EXIT, and return to the Board options; select (F10), EXIT, and move up to the subroutines; and then select (F10) again to move up to the nine major or top level options. Note that if you select (F10), EXIT, while

in the top or major routine command window, the following message appears:

**!MAKE SURE YOU SAVED JOB TO DISK! OK TO EXIT PROGRAM Y(N)?**

This message is presented to remind that you save the design that you have been working on before leaving the program.

There are two "pop-up" windows associated with the CRT. The first is an Information window that contains GRID, Layer, Units, Job Name. The Window is toggled on by CTRL-I. A second CTRL-I will add an addition to this Information window, containing the database or board outline (white) and the area displayed at the CRT at any time in a solid color. This feature is useful for orientation purposes when designing large boards. Alternately, Part Name, Connection, Line Width, etc. is shown. Note that not all the information is displayed at any one time. Instead, the information pertinent to the current activity is displayed. For example: When moving a part, the part data is displayed.

The second window is a "Location" window and is displayed On/Off with CTR-L. X Y indicators represent the location of the cursor relative to the system origin. DX and XY represent the distance the cursor has moved from a start of a line or from the last corner when drawing a line or polygon. The X & Y indicator always updates in system grid units, i.e. .010, .025, etc.

PRACTICE: At this time, take a few minutes out to select and study each of the nine major routines. Study them carefully and become familiar with at least the subroutines associated with each of the major routines. Additionally, turn the two pop-up windows ON-OFF.

THE PAYOFF!: To become an exceptionally talented designer using the PADS-2000 program, you must develop skills with the Command Menu and its hierarchical implementation. You must be able to use the Function Keys and Mouse much the same as a skilled typist uses a keyboard. In actuality, this is not difficult and, in fact, will become quite easy with practice. For the time being, just start your acquaintance with the command menu hierarchy and its usage and commit as much of it to memory as possible.

## 2.3 INPUT/OUTPUT

We will now become well acquainted with the first major option: IN/OUT.

Select F1, IN/OUT either with the F1 Function Key or the Mouse. Study the subroutines associated with the Input/Output routine.

One of the functions of the routine is to save or store your work on the hard disk and recall it, using the Job In and Job Out subroutine. (These will be the subroutines used in this practice session.)

In addition to the Job In/Job Out subroutines, the ASCII IN and ASCII OUT subroutines are used to create ASCII files (ASCII OUT) or accept ASCII files (ASCII IN), that in turn can be used to interface the PADS-2000 software to PADS-LOGIC or other systems, or input manually prepared schematics, using the ASCII database format.

The FutureNet In™ subroutine allows the inputting of a schematic generated in a FutureNet format to be inputted into the PADS-2000 program for the design of the board.

The CONV IN accepts board databases designed on a Redac™ System and converts the database into PADS-2000 format.

The Schema™ In accepts data prepared on the Schema Logic Capture System.

The On-The-Fly option permits entering Parts and Connections at the CRT, without using Connections and Parts Lists or Schematic Input.

The Erase Job option permits a file that is currently in memory and displayed at the screen to be erased.

Let's now get some hands-on experience with the In/Out option.

We have prepared for your use, a series of files that contain the Connections, Parts and Board Outline of the board you will design in the Practice sessions. The schematic representing the board is shown in Figure 2.3, the Parts List in Figure 2.4, the Board Outline in Figure 2.5 and the Design Criteria in Figure 2.6. Take time now and study these

four figures for a few minutes. (These figures are located at the end of this chapter.)

Figure 2.7 represents a finished designed board that you will use to practice INPUT/OUTPUT and SETUP.

The name of the completely designed practice job file is TPRACTIC.JOB and it is located on the hard disk in the FILES directory. We will now practice retrieving this file from the hard disk, bringing it in memory, and displaying it at the screen just as if you had originally created it. (Don't concern yourself at this point as to how this file was created--your effort during the training sessions will produce the same results!)

Be in the highest order of major options and select (F1), IN/OUT.

When in IN/OUT, select (F1), JOB IN.

In response to the message asking for the Job Input File Name, type in:

**TPRACTIC<CR>**

NOTE: <CR> at the end of a text string means select the ENTER Key to enter the data.

NOTE: Although the name of the file that you are recalling is TPRACTIC.JOB, you need not enter the .JOB extension. You need only enter the primary name of the file, i.e., TPRACTIC.

The file will be read from the hard disk into memory and displayed at the screen.

NOTE: If you have the Information pop-up window displayed you will see "Working xx". This "Working" is always displayed when the CPU is engaged in some activity, computing, making a file, saving, etc. If during a long, long, sequence, you become concerned that your CPU may be locked up, check the "Working xx" message; if it is present, the CPU is continuing to process.

Suppose you could not remember the name of your Input file, you could:

1) Exit to DOS, enter the FILES Directory, use DIR and get a listing.(awkward)

OR BETTER:

2) In response to the message asking for the JOB INPUT FILE NAME, type:

*** <CR>**

A pop-up window will appear containing the file names. Move the cursor over TPRACTIC.JOB and select mouse button One. TPRACTIC.JOB will be read into memory and displayed at the CRT.

## 2.4 THE SETUP ROUTINE

When SETUP is selected, the following subroutines are displayed.

| | | | | |
|---|---|---|---|---|
| **1 Display** | **2 Pads** | **3 Sizes** | **4 Params** | **5 Net Attr** |
| **6 Macros** | **7 Sr Parms** | **8 Units** | **9 Origin** | **0 Exit** |

SETUP contains options used to customize or tailor the system database for the board being designed as your personal desires. An important characteristic of SetUp is that when the Default settings for the various system features, as shipped with the software again have been changed locally for the board using SetUp, these new Default conditions remain "as set" with the board even during storage, until changed again. The original Default settings, however, do not change within the software; thus when a new board is started, the original Default settings are available and will remain until changed for the new board by a new SETUP selection.

Select SETUP, (F2).

Note that there are nine different items in SETUP and we will now spend some time with them.

## 2.4.1 DISPLAY

DISPLAY (F1) is used to make various items visible or invisible at the screen. If the items are made visible, a choice is given as to the color to be used on the item. Let's get acquainted with Display by selecting (F1), DISPLAY.

The screen presentation changes drastically, therefore, let's familiarize ourselves with its contents. The top line contains a color palette of 16 colors. An item can be identified by color. "Items" consist of connections, 2-D lines, lines, text, tracks, copper, vias, errors, top & bottom layer, Part shapes, Part names and Part types. Associated with all of these items except for part data and connections are the numbers from 1 to 30. These numbers represent Levels or Layers in the database. Level 1 is always used for the level or layer nearest to you at the screen, with the increasing number of levels up to 30, going away from you. Generally, track and ground/power planes artwork utilize the lower number levels while documentation such as assembly drawings, silkscreens, pads only, etc. utilize the higher levels. In this particular practice example, we will be using Levels 1 and 2 for the artwork of the board, level 24 for a Drill Drawing, Level 28 for a Pads Only Solder Mask layer, either Level 26 or 29 for Silkscreen and Level 27 for an Assembly Drawing.

We have already set up DISPLAY on the Training file for you for these conditions, however, let's get briefly acquainted with how this was done.

Move the cursor over a color that you wish and select with the left mouse button. The rectangle at the center top of the screen displays the color you have selected. Now bring the cursor down over a level (Layer) of an item and click Mouse button #1 and notice that the level number is boxed by the color you have selected. EXIT (F10) and notice that the item whose level and color you changed has indeed changed to the color desired.

To make an item invisible, place the cursor over the same color block as that of the CRT background (in the instance of TPRACTIC.JOB it is Dark Blue) and select. Notice that the large rectangle at the top of the screen goes Blue (invisible). Then move the cursor over the level of the item to be made invisible (Tracks Level one), and Select. Notice the box

surrounding the level number disappears and upon exit, the tracks on Level one are invisible.

Lesson: To make an item invisible, make it the same color as the CRT background color.

Following this, make tracks, Level 1 yellow again (visible).

Note that at the bottom of the screen, the items: Top, Bottom, Part Name, and Part Type. Top is the component outline on the top layer; Bottom is the component outline on the bottom layer; Part Name is the Reference Designation, and Part Type is the type of the Part. Part Name and Part Type can be displayed and plotted as part of your Assembly Drawing.

TOP is used to display the part outlines on the top layer.

BOTTOM is used to display the part outlines of parts located on the bottom layer.

The E is for an "Electrical" Layer, i.e., a layer that has parts on it.

For Part Outlines to be displayed on the bottom layer, the parts must be located on the bottom layer.

The two layers associated with Part Outlines are the top (front) and bottom artwork layers. The rational for allowing you to display and plot part information on front side and back side of a board is to handle those situations where you may have Surface Mounted Devices and other Parts on both sides of the board. To display, choose the colors and select the E for either TOP or BOTTOM as appropriate.

Unrouted connections use a 0 to indicate not on a layer and they are made visible or not, by the selection of the appropriate color to make the connections visible.

NOTE: For an item to be space checked, (in Copper, Text, Tracks, Via, etc.), it must be displayed (Visible) using SETUP.

Suppose you do not like blue as the CRT background color - suppose you like black. Do the following: Select Black and change BakGnd to Black and EXIT. The CRT background is now Black.

Work with the display until you are comfortable with DISPLAY.

## 2.4.2 PADS

PADS is used to change the size, shape and drill size of Pads or Pad Stacks on a Part Decal. The selection of LIST ALL (F1) will result in a printout or listing of the information on all Part Decals on the board. The entry of a Decal Name will result in the Pad Stack data of that Decal being displayed at the CRT. Try it with the Decal DIP14.

A response to the prompt:

**NAME OF PART DECAL**

with the name of a Part Decal will result in a Text Window at the CRT, listing the information on top, bottom and middle layers of the parts of the Named Decal. The pad information may be changed using this Text Window.

## 2.4.3 SIZES

The SIZE subroutine (F3) is used to setup the size of various items on the board. When selected a text window is displayed showing the various items and then size as contained in the board currently in memory. The white numbers are the "default" numbers that an item takes when a database is initially created. The Aqua color shows the size of an item as it is currently in the database.

Select SIZE (F3). Study the text window. Note the Aqua text sizes are the same as the default setup. This is because the file TPRACTIC.JOB has taken the default values and has not been changed. Let's make some changes. Let's change the board outline width to 10 mils. Enter 10 <CR> and the Aqua color changes to 10. The database board outline is now set to 10 mils and will remain at 10 mils until changed again.

Notice there are two different width tracks on the TPRACTIC board. 50 mils for Power and Ground and 12 mils for Signal .

Exit SIZES and redisplay the board.

Here is an exercise. Expand (Numeric Key 9) until you can clearly see the width of the 12 mil signal tracks. Select SETUP, SIZE and change the tracks 12 to 25. Exit and examine the 12 mil tracks. Note that they are now double this original width at 25 mils.

### 2.4.4 PARAM(ETERS)

PARAMETERS is similar to SIZES, in that it is a text window used to set various parameter of the board-under-design database.

#### SHOW CURRENT LEVEL

If the option is in the Off condition, priority is given to displaying the item on a level in a 3-D fashion; that is, Level 1 items first, then level 2, then level 3, etc. For example: A Pad Stack with pads on Levels 1, 2, 3 and 4 will have Level 1 shown, the other 3 levels are hidden behind the Level 1 pads. If a track on level 4 occupies the same position as a track on Layer 1, it will be hidden under the track on Layer 1.

A designer may wish to give priority to the Level on which routing a track is in process. For example: if routing on Layer 2, the designer may wish the items (pads, vias and tracks) on Layer 2 to be given priority over items on other levels. SHOW CURRENT LEVEL is the means by which the current level of routing is given priority over items on other levels. Toggle the OFF to On.

In the above example, when the designer goes to Level 2 from another Level with "Current Level Last ON," the display will repaint, giving priority to Level 2 items. That is to say, the pads, vias and tracks on Layer 2 are shown, and the pads, vias and concurrently located tracks will be behind and hidden behind those on Layer 2. This feature is very subjective and you need to try both ON and OFF before deciding which you prefer.

#### CURRENT NET HIGHLIGHT

The selection of ON highlights the net selected for interactive routing.

TEAR DROP PAD GENERATION

ON generates a Tear Drop Pad at the junction of a track and a Round pad. Try it. Switch it ON and examine the tracks of TPRACTIC.JOB. (You will need to expand the CRT--use the PGUP key.)

COPPER POUR HATCH DISPLAY

An ON results in the "poured" area of copper being displayed as an X-Y hatched area instead of a blank outline.

DOT GRID

Determines the size of the Dot Grid on the CRT Working area.

REAL WIDTH

Permits display of tracks at either their real width or center line only. All tracks at or above the Real Width setting are displayed at their actual width. All below the real width will have their center line location only displayed.

Let's try this illustration. The tracks on the "practice" board are at 50 mils and 12 mils. Set Real Width at 100 and note that all tracks are at their center line. Next, set Real Width at 50 and then Exit and observe the Power and Ground tracks at their Real Width, and Signal connections at a center line width. Next, change Real Width to 12 and observe the Display (you will need to enlarge an area to see the 12 mil tracks shown at their Real Width). All tracks on the board will now be at their Real Width.

NOTE: It is also possible to set Real Width with the Modeless command "RX" where X is the Width above which the Real Width is to be displayed.

THERMAL LINE WIDTH

This parameter defines the line width to draw the Thermal Relief line which tie into the copper pour plane. Changing this value after a copper area has been flooded required the selection of FLOOD again.

COPPER HATCH GRID

Establishes the separation of lines of copper, as used by the photoplotter to Cross Hatch Poured Copper. Used in conjunction with aperture size when drawing copper (set in the Routine SIZE, see 2.4.3) to determine whether copper area will be hatched or solid. A Copper Line of 50 mils and a Copper Hatch Grid of 50 would result in a solid copper area; a Copper Line of 10 mils and a Copper Hatch Grid of 50 mils would result in a screen mesh (hatched) area.

NOTE: The "finer" the resolution of the copper pour, the more detailed the copper poured area. This may not be desirable in that a fine grid size of less than Copper Grid = 25 will result in isolated polygons of copper when used in areas that contain tracks.

COPPER POUR CLEARANCE

Establishes the space between "poured copper" and isolated pads and tracks within the poured copper area.

CLEARANCE (P/P, P/T, T/T)

Establishes the clearance between pad and pad, pad and track and track and track. Used in the Checking program.

BACKUP INTERVAL

BACKUP is a means of automatically saving your files for use in the event of an emergency or power out, etc. What happens is at the time of the backup time interval (in this instance, 5 minutes), and without action on your part, the database as it exists at that time is stored on hard disk as a special backup file for your use in the event that you need to use it. Each time a new backup file is made, the preceding backup file is overwritten, thus you always have access to the preceding 5 minutes of work in event of need. The five minutes used as the backup job file interval could be changed by entering the number of minutes for the new interval. We recommend 5 minutes as the optimum time, however, for taking backup jobs, and we recommend that you do not change the backup job interval time.

NOTE: Should your system "hang-up," "crash," or have a power failure, you can recover your work without significant loss; Reboot the

system, and in response to JOB IN of the IN/OUT routine, strike the <CR> Enter key and the last backup file made will be read in and displayed, ready for you to pick up your design effort, with only five minutes of your time lost!

NOTE: The back-up interval is NOT observed when you are at a low subroutine level; MODIFY ROUTE within the ROUTE option, for example. You should therefore exit from these lower level subroutines every half hour or so, in order to force a back-up file.

MAX ROUTING LEVEL
OLD TO NEW LEVEL RE-ASSIGNMENT

These two entries function together to create a multi-layer board out of the 2-Layer Default board established at the time of the entry of database input data. To use, in response to:

**MAX ROUTING LEVEL**

enter the maximum number of layers planned. Following this, reassign the layers in the 2-layer board to the appropriate layers in the multi-layer board.

Example: Converting a 2-layer board to a 4-layer board. Assign old Layer 1 to new Layer 1; assign old Layer 2 to new Layer 4. This results in a 4-layer board with any routes and pads on Layer 1 remaining on Layer 1, and any routes and pads on Layer 2 being transferred to Layer 4. Thru vias penetrate all four layers. New layers 2 and 3 may be used for either routing layers or Pwr/Gnd Planes.

NOTE: When creating a multi-layer board, you MUST set the colors in SETUP, DISPLAY to the appropriate color for each layer, in order to be displayed at the CRT.

NOTE: Although you can increase the number of layers at anytime you wish in the board design, from 2 up to a total of 30, once increased, these layers cannot be decreased. Be aware of this if you ever, during the course of design of a board, wish to change to a multi-layer board, you may increase the number of layers but you may not decrease the number of layers.

NOTE: When changing from a 2 layer board to a multi-layer board, be sure the Pad stacks for Parts and Vias are set with a Size and Shape on all layers.

### 2.4.5 NET ATTR(IBUTE)

NET ATTRIBUTE is used to assign certain net attributes to all nets on the board or specific nets. These attributes are as follows:

Display - Turns Net OFF for (1) Display purposes, (2) Length Minimization in Placement, and (3) Autorouting.

Nets whose Names have been individually selected using ADD ITEM (Not ALL), have two additional display conditions, in addition to ON and OFF. These two conditions are PR1 and PR2. These latter conditions allow you to control Partial Route, i.e., those connections that are not completely routed, but are routed "partially" with a part left as a connection.

PR1 results in a Partial track being visible, and its connection being made invisible, the remainder of the Net will be visible.

PR2 results in the track portion or Partial track of the connection being visible and all the remainder of the connections in the net being made invisible.

Length Minimization - Used to set the type of minimization used in Auto Parts Placement. Three types are available:

- Minimum - Algorithm calculates true minimum length of a net
- Vertical - Algorithm acts to bias connection in a Vertical direction and calculates resulting minimum length
- Horizontal - Algorithm acts to bias connections in a Horizontal direction and calculates resulting minimum length.

ECL - No Length Minimization performed on the net, left "as is"

Share - Permits sharing of copper between two connections within a net during Autorouting with SHOVE and SUPERROUTER.

Plane - The net is NOT routed in Autorouting. Use when the Net is to create Thermal Pads on the appropriate plane. A YES tags the Net as a "Plane" net.

Ripup - Used to flag a net for "ripping up" in the SHOVE and SUPERROUTER Autorouter, to make room for another track being routed. Use NO only on those nets whose track position is critical and thus should not be ripped up by the Autorouter.

Shove - Used to flag a net for "No Shove" in the SHOVE or SUPERROUTER Autorouter, to make room for another track being routed. Use NO only on those routes whose position is critical and thus should not be shoved by the Autorouter.

The Default of NET ATTRIBUTES is ALL connections. To change an individual net to another set of Attributes, use the ADD ITEM subroutine and enter the name of the net ; then set its Attributes as required.

NOTE: Setting a Net to OFF in NET ATTRIBUTES, DISPLAY disables the Net from being displayed, routed or checked.

AN EXERCISE: Set Power to Display Off and Vertical Minimization by selecting ADD ITEM, typing in VCC, then turning VCC Display to Off.

## 2.4.6 MACROS

Do Not, at this early time in your Self-Training program, involve yourself in MACROS. After you are familiar with all the Design features, then study MACROS. Refer to a sperate chapter--Chapter 11, MACROS.

## 2.4.7 SR PARMS

Used to set up parameters for the PADS-SUPERROUTER (You must have the SUPERROUTER option for this subroutine to be active.

Routing Grid - Toggles between 10, 12.5, 16.67, 20, 25 and 50 mil grid.

Default Track Width - The track width to be used by the Superrouter.

Minimum Track Clearance - The clearance between the tracks being autorouted and other items

Board Edge Clearance - The "keep clear" from edge of board distance in mils.

Off Grid Track Corners - A Yes will result in all tracks corners at 45 degrees. A No will have a mixture of 45 and 90 degrees.

Via Shift Allowed - Yes/No (on 16.67 Grid Only)`

Level Direction Flags - Sets the major routing direction for each layer. Choices are (1) Vertical, (2) Horizontal, (3) Any and (4) None.

NOTE: A NONE will prevent routing from occurring on that layer.

### 2.4.8 UNITS

The UNITS command lets you select the dimension for PADS-2000 to be either inch or metric. There are three choices: Metric, Inch, or Thousandths of an Inch. When selected, respond to the prompt:

**Units are Metric, Mils or Inches [Mils] (ME/MI/I)? >**

Type:

**ME<CR>**

for Metric,

**I<CR>**

for Inch,

**MI<CR>**

for Mil or 1 Thousandths of an Inch.

When Metric is selected, all of the display values convert to metric. The basic unit is millimeters and the possible resolution is .001mm. Your input will be in millimeters, for example, 1.345 mm is input as 1.345 <CR>.

When Inch is selected, the unit is inches, and the resolution is .00001". Your input will be in inches, for example, 1.645" is input as 1.645 <CR>.

When Thousandths of an inch is selected, the unit is .001", and the resolution is .00001". Your input will be in .001", for example, 1.645" is input as 1645 <CR>.

You must remember to enter all dimensional values requested by the system using the current coordinate system. You are free to move between metric inch and thousandths of an inch coordinate systems at any time in the design. The fundamental system grid insures that there is no loss of data accuracy when you switch coordinate systems.

NOTE: When items are stored in a file, the database is stored in metric units, regardless of whether the entry was in Inch or Metric. This does not affect the display of data.

Set UNITS to Mils.

## 2.4.9 ORIGIN

You will recall in the early discussion of the Screen Presentation that the system origin was located at the lower left corner of the database (and the lower left corner of the screen). That is true until the Origin has been moved. One normally moves the Origin from the lower left hand corner of the database to be co-located with the 0,0 reference of a board outline when creating the Board Outline. To move the Origin, place the cursor at the new location and select ORIGIN (F9).

In the TPRACTIC.JOB file the Origin is located at the lower left corner of the board (the white circle). Try moving the Origin to the upper left corner. Place the cursor over the upper left corner and select ORIGIN (F9). Note the change in coordinate reference.

Caution: When the Grid value is changed and the Origin is relocated, the System Grid, including the Placement and Routing Grid, will be shifted. This situation, if not allowed for, may be disruptive to the Automatic Router. Further details will be provided during the discussion on Routing (Section 4.5.5).

## 2.4.10 SCREEN MANIPULATION

As part of this first Tutorial, we wish to introduce you to manipulating or moving the design around on the screen. This is accomplished with the Numeric Key pads on the right hand side of your keyboard and the cursor, or by using certain routines controlled by the Function Keys. (The TPRACTIC board should be displayed at the screen for this exercise.)

NOTE: THE NUMERIC KEY LOCK MUST BE OFF

Numeric Key Pad 1 - Repaint. This key pad does not move the display at the screen, instead it refreshes the CRT presentation.

Numeric Key Pad 7 - Reset. This resets the display such that if there is a Board Outline in the file at the time, the database will be rescaled so that the Board Outline fits inside the screen and the contents are displayed. If there is no Board Outline, the entire database is displayed after Reset. In general, Reset is used when you have windowed around on the board or expanded to the point that you are not quite sure where you are on the board, and you want to Reset the board for reference purposes.

DEL (Delete) BOX WINDOW This key is used to change the scale of a section of the board so as to "enlarge it" as required. To use, select the DEL key, place the cursor on the lower left hand corner of the area to be magnified and select mouse button #1. Then move the cursor upward to the right until the rectangle being generated by the cursor, encloses the area to be magnified. Then select mouse button #1 again. The area within the rectangle will have its scale changed such that the screen window is filled by the area enclosed by the rectangle.

To reset Box Window, use Numeric Key 7, Reset. As a short practice session, practice the Box Window and Reset until you are comfortable with the two commands.

PAN - Numeric Keys

These keys are used to move or pan the presentation at the CRT around to different locations of the CRT. In the process, they do not enlarge or contract the scale. The algorithm functions as follows: The database content located at the edge of the CRT at the time a Pan key is struck, is relocated such that it is centered at the CRT.

NOTE: The CRT image appears to move in the direction opposite to the Pan arrow.

To PAN, press Numeric Keys 2, 4, 6 or 8, depending upon the direction you wish to pan. The scale will remain unchanged but the feature that was at the side of the CRT will be repositioned approximately to the center of the CRT. This feature is particularly useful when interactively routing a track and you progress to the edge of the CRT; striking a PAN key will move the track that is at the edge of the CRT to the center of the CRT where you can continue the Interactive routing process.

EXPAND - Numeric Key 9 (PG UP). Moves that feature of the board under the cursor at the time of expansion to the center of the screen and multiplies the scale of that area such that the features are doubled in size.

CONTRACT - Numeric Key 3. Moves the feature under the cursor at the time of contraction to the center of the screen and changes the scale to reduce the size of the item being displayed by one-half (more of the board area is displayed).

CENTER - Numeric Key 0. Acts to bring the area under the cursor to the center of the CRT. Scale is not changed.

Note the Postage Stamp feature in the small Information Window. It contains the board outline (or database working area if no board present). A solid Red Postage Stamp represents the part of the board displayed at the CRT. EXPAND the area displayed, use PAN and note the Postage Stamp action.

TAKE A PRACTICE SESSION: We suggest that you take five-ten minutes now and practice on the screen manipulation commands using the TPRACTIC board. Pan can be particularly interesting until you have gotten a good feeling for the algorithm and can anticipate where the item will be located after the Pan, Expand, Contract and Window commands are completed. Practice each of the screen manipulation routines until they become second nature and you can predict with a fair degree of accuracy where the item will be on the screen after the manipulation is completed.

### 2.4.11 MODELESS COMMANDS

Let's get acquainted with the concept of "Modeless Commands" or commands that are entered at the Keyboard without using the Mouse. Take time out to go to Volume II, Chapter 2, Section 2.4. Read this short section and practice some of the commands.

## 2.5 JOB OUT

We wish to continue your training on the IN/OUT Option. In the beginning of the Tutorial, you brought the job "TPRACTIC.JOB" into the system using JOB IN and now before we depart from IN/OUT, we wish to show you how to Save your work files, during and after work.

Go to the main menu of options. Select IN/OUT (F1). Now select Job Out.(F2). In response to the message:

**JOB OUTPUT FILE NAME**

type in any name up to 8 characters in length. (Don't use a space in a file name since a space is illegal.) Immediately after you use the Enter key, note that the Information window displays "Working" for a second or two. Your file is being written on the disk and upon the completion of "Working," exists on the hard disk under the name you gave it.

VERY IMPORTANT! You need to make a habit of creating a Job Save log with each board that you design. Consequently, for each board, make a Job Save Log similar to that shown below. Enter the name of the file, when it was made, who made it and a short description of the file. Hereafter, in these self-training tutorials, each time you make a

work file of your Practice job, enter it in your Job Save Log. This will help you greatly if you have to go back to some point in your design and pick up and start again, in that you will know exactly what file to go to and the contents of that file.

| JOB SAVE LOG | | | | |
|---|---|---|---|---|
| Job Name | Date | Time | Designer Initials | Job Description |
| | | | | |

In the IN/OUT routine you no doubt have noticed ASCII IN (F3) and ASCII OUT (F4). These subroutines are used to bring files prepared in the system Text Editor PEDIT, into memory (ASCII IN) or to convert a binary database of the job currently in memory, into ASCII (ASCII OUT) and store it on hard disk. In the next Tutorial you will retrieve your Training board Connection List and Parts List, that have been prepared in an ASCII format and stored on the hard disk for your use during the training sessions.

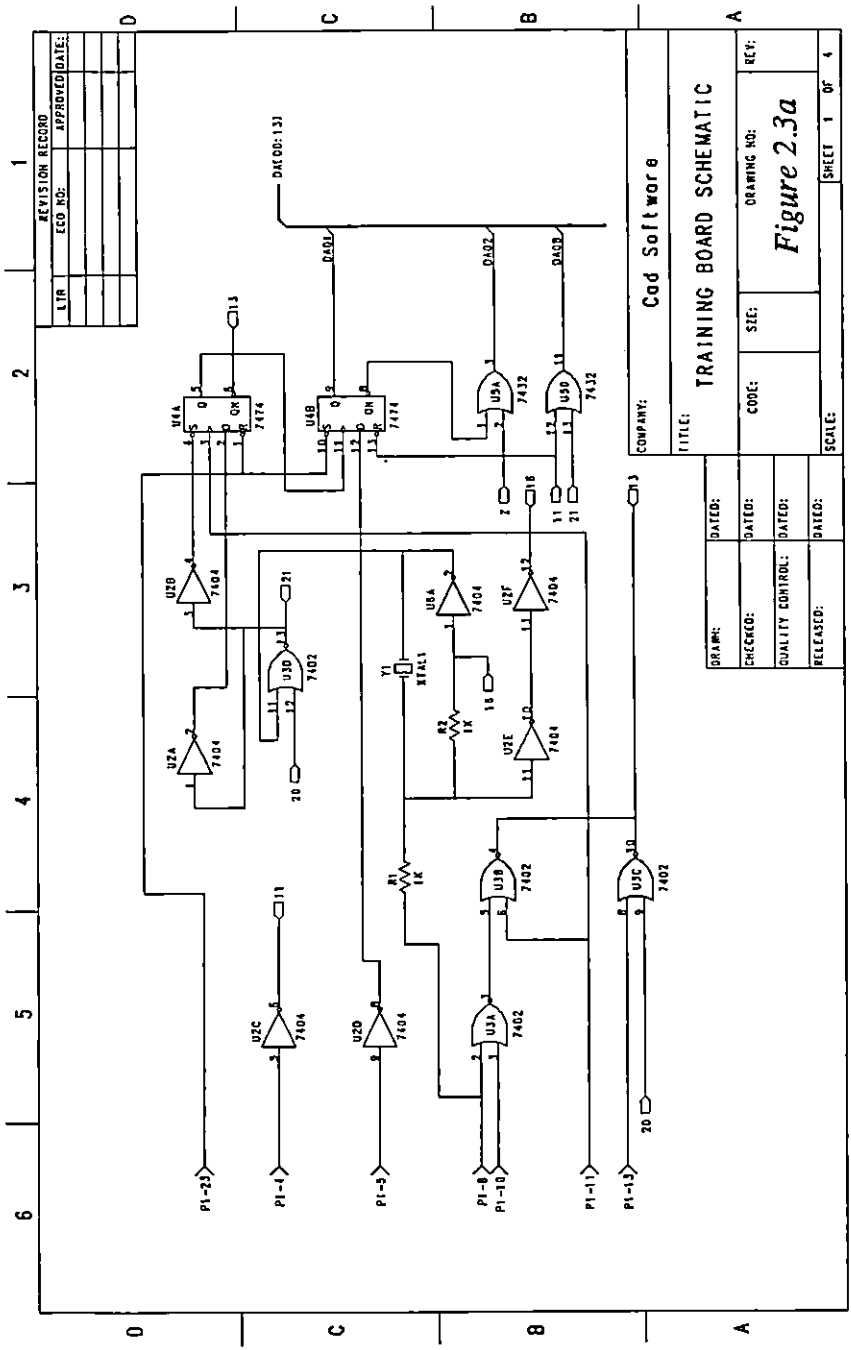
The Input option of Design-On-The-Fly will be covered in Chapter 7, Tutorial 6.

FutureNet, Schema and CONV will not be covered in the Tutorials, however, complete details are included in the Input/Output Chapter of Volume II Reference Manual.

This completes Tutorial 1.

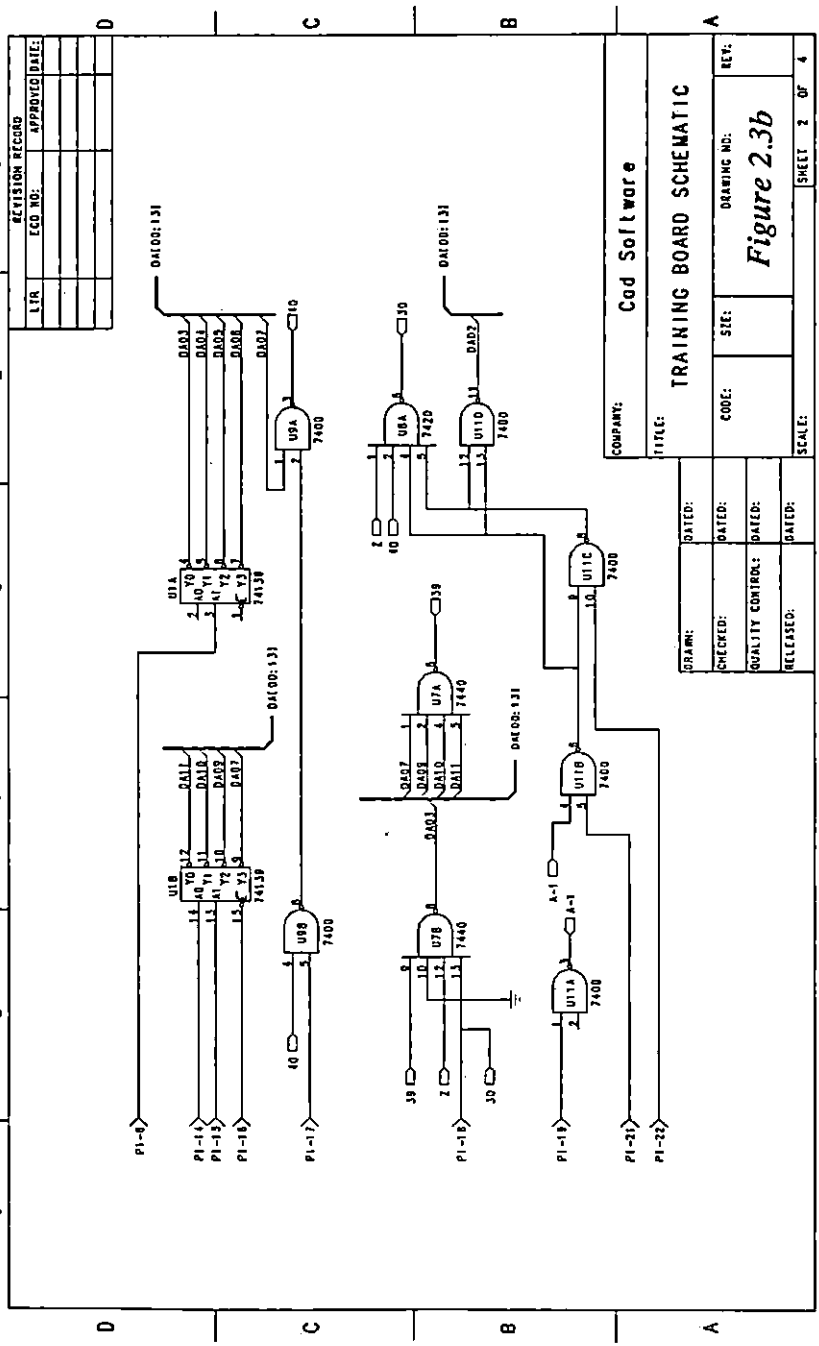
You should be fairly confident of your knowledge of the contents of the screen display, how to call up Jobs, as well as to write them down to disk, how to SetUp the database at the beginning of the design of the board, etc.

**REMINDER**:
HAVE YOU SENT IN YOUR REGISTRATION CARD?
HAVE YOU PURCHASED THE EXTENDED SUPPORT CONTRACT.

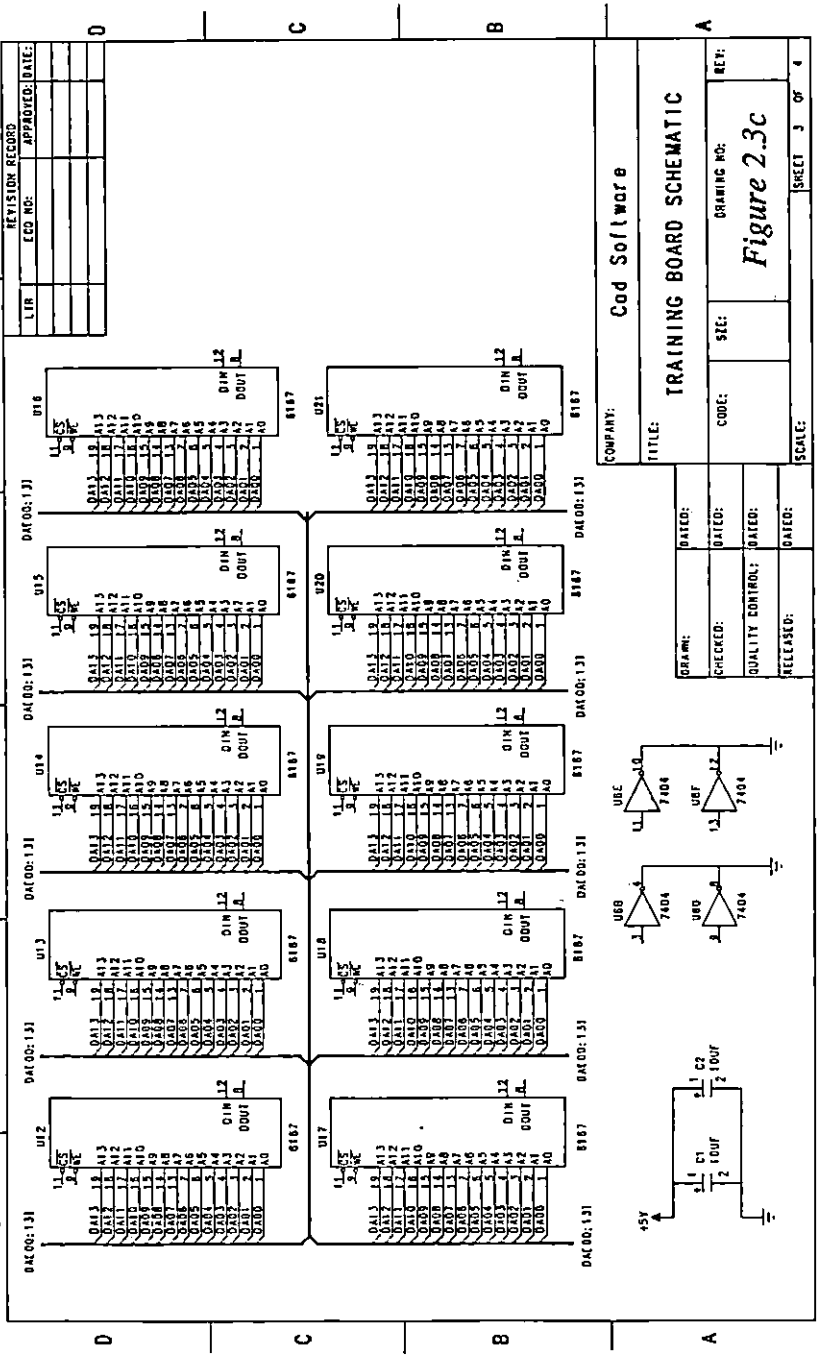



*TRAINING BOARD SCHEMATIC, SHEET 1*

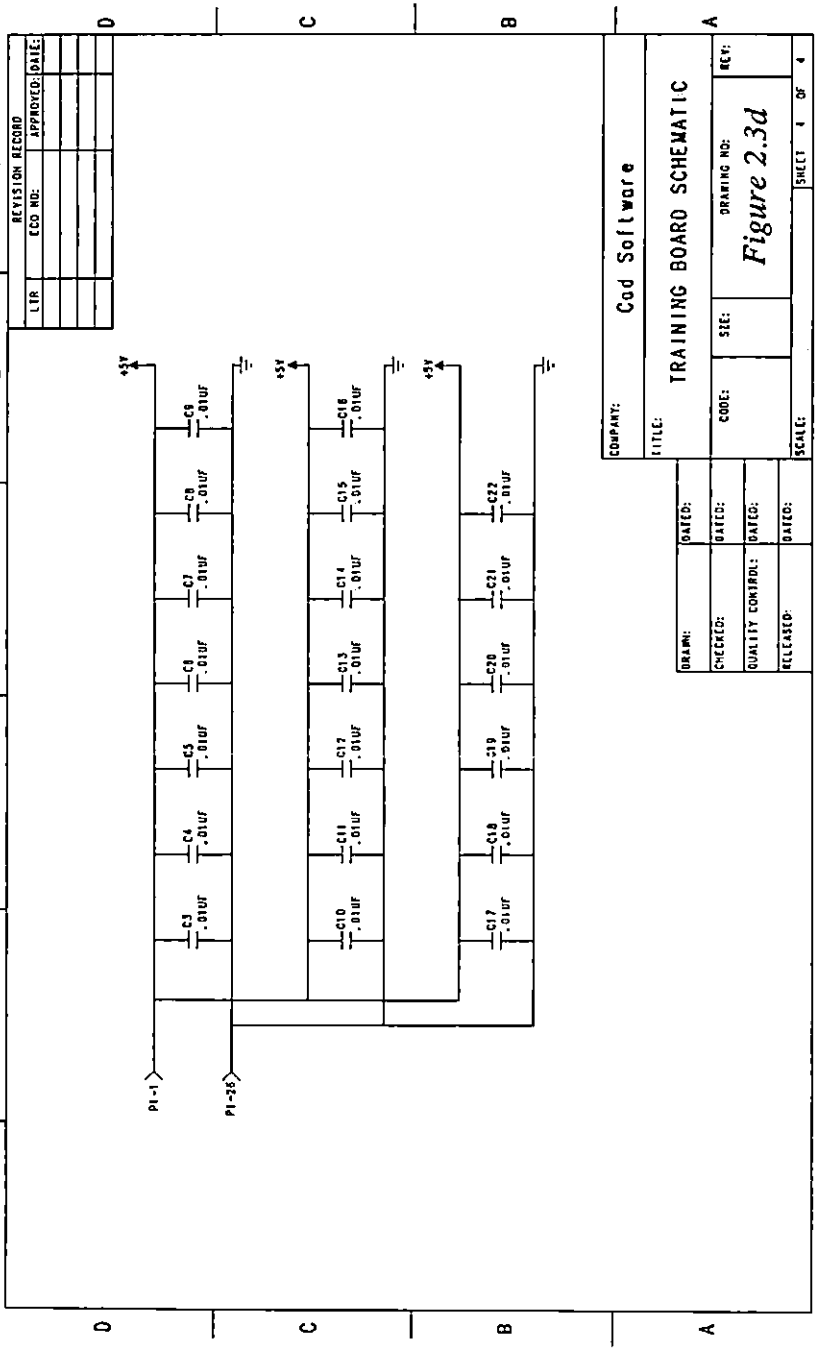
*Figure 2.3a*




*TRAINING BOARD SCHEMATIC, SHEET 2*




*TRAINING BOARD SCHEMATIC, SHEET 3*




*TRAINING BOARD SCHEMATIC, SHEET 4*

## TRAINING BOARD PARTS LIST

*PADS-2000* FILE INFORMATION

| *PART* | ITEMS |
|---|---|
| U2 | 7404@DIP14 |
| U3 | 7402@DIP14 |
| U4 | 7474@DIP14 |
| U5 | 7432@DIP14 |
| P1 | CON\26P\ED@CON\26P\ED 700 -300 |
| Y1 | XTAL1@XTAL1 |
| R1 | R1/8W,1K@R1/8W |
| R2 | R1/8W,1K@R1/8W |
| U6 | 7404@DIP14 |
| U1 | 74139@DIP16 |
| U7 | 7440@DIP14 |
| U8 | 7420@DIP14 |
| U9 | 7400@DIP14 |
| U11 | 7400@DIP14 |
| U12 | 6167@DIP20 |
| U13 | 6167@DIP20 |
| U14 | 6167@DIP20 |
| U15 | 6167@DIP20 |
| U16 | 6167@DIP20 |
| U17 | 6167@DIP20 |
| U18 | 6167@DIP20 |
| U19 | 6167@DIP20 |
| U20 | 6167@DIP20 |
| U21 | 6167@DIP20 |
| C1 | CL25,10UF@CL25 |
| C2 | CL25,10UF@CL25 |
| C3 | DCAP1,.01UF@DCAP1 |
| C4 | DCAP1,.01UF@DCAP1 |
| C5 | DCAP1,.01UF@DCAP1 |
| C6 | DCAP1,.01UF@DCAP1 |
| C7 | DCAP1,.01UF@DCAP1 |
| C8 | DCAP1,.01UF@DCAP1 |

```
C9      DCAP1,.01UF@DCAP1
C10     DCAP1,.01UF@DCAP1
C11     DCAP1,.01UF@DCAP1
C12     DCAP1,.01UF@DCAP1
C13     DCAP1,.01UF@DCAP1
C14     DCAP1,.01UF@DCAP1
C15     DCAP1,.01UF@DCAP1
C16     DCAP1,.01UF@DCAP1
C17     DCAP1,.01UF@DCAP1
C18     DCAP1,.01UF@DCAP1
C19     DCAP1,.01UF@DCAP1
C20     DCAP1,.01UF@DCAP1
C21     DCAP1,.01UF@DCAP1
C22     DCAP1,.01UF@DCAP1

*END*   OF ASCII OUTPUT FILE
```

*Figure 2.4 - TRAINING BOARD PARTS LIST*

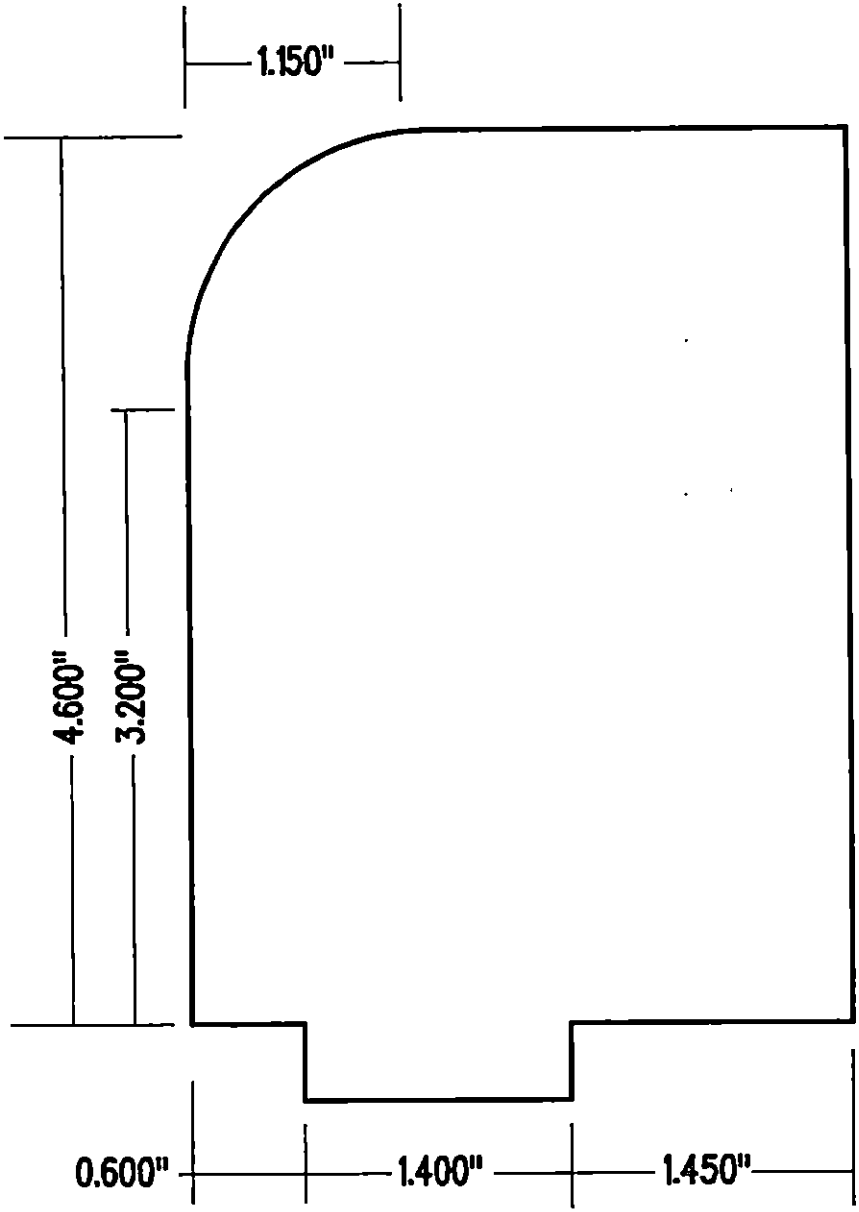



fig 2.5 Board Outline

# DESIGN CRITERIA

1) 2 Layer Board, Tracks Layer 1, Horizontal Layer 2, Vertical
2) Signal Tracks - 12 mil, VCC and GND 50 mil IC pads - 60 mil Round, 37 mil Drill Analog Components - 60 mil Round, 37 mil Drill
   Vias - 55 mil Round, 37 mil Drill
   Edge Fingers - 75 x 400 mil, Space at 100 mil
3) Locate IC's Vertically on board, with Pin 1 in upper left corner
4) Place your name in .150" high Text at top of board, Layer 1
5) P1 is 26-pin Edge Finger Connector, Odd pins Layer 1, Even pins Layer 2. Locate Pin 1, Edge Finger .700" in X and -.300" in Y from Board Origin
6) Keep Components and Tracks at least .01" from board edge
7) Place Filter Caps C1 and C2 close to the Connector P1
8) Place Y1, R1 and R2 at the top of the Board outside the Digital area. Place a Ground Plane on Layer 1, under the Parts.
9) Locate Memory IC's at top of board.
10) All spacing except copper to be 13 mil. Copper clearance to be 15 mil
11) After parts are placed, rename IC parts with lowest number component in upper left corner, and highest number parts in lower right corner of board.
12) Place the Part Names, centered under the Component Outline and reading right to left.

*Figure 2.6 - DESIGN CRITERIA FOR TRAINING BOARD*

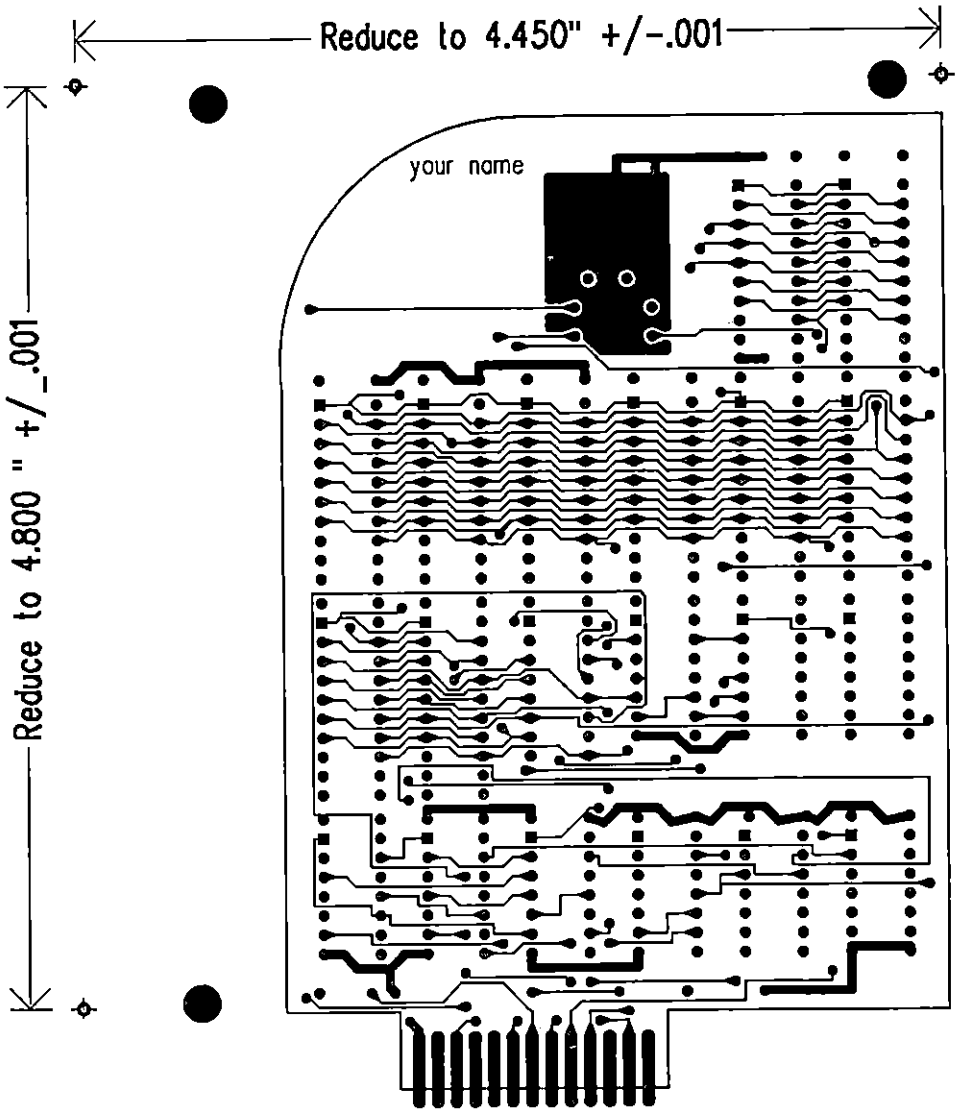



Figure 2.7a

*Figure 2.7a - FINISHED TRAINING BOARD, LAYER 1*

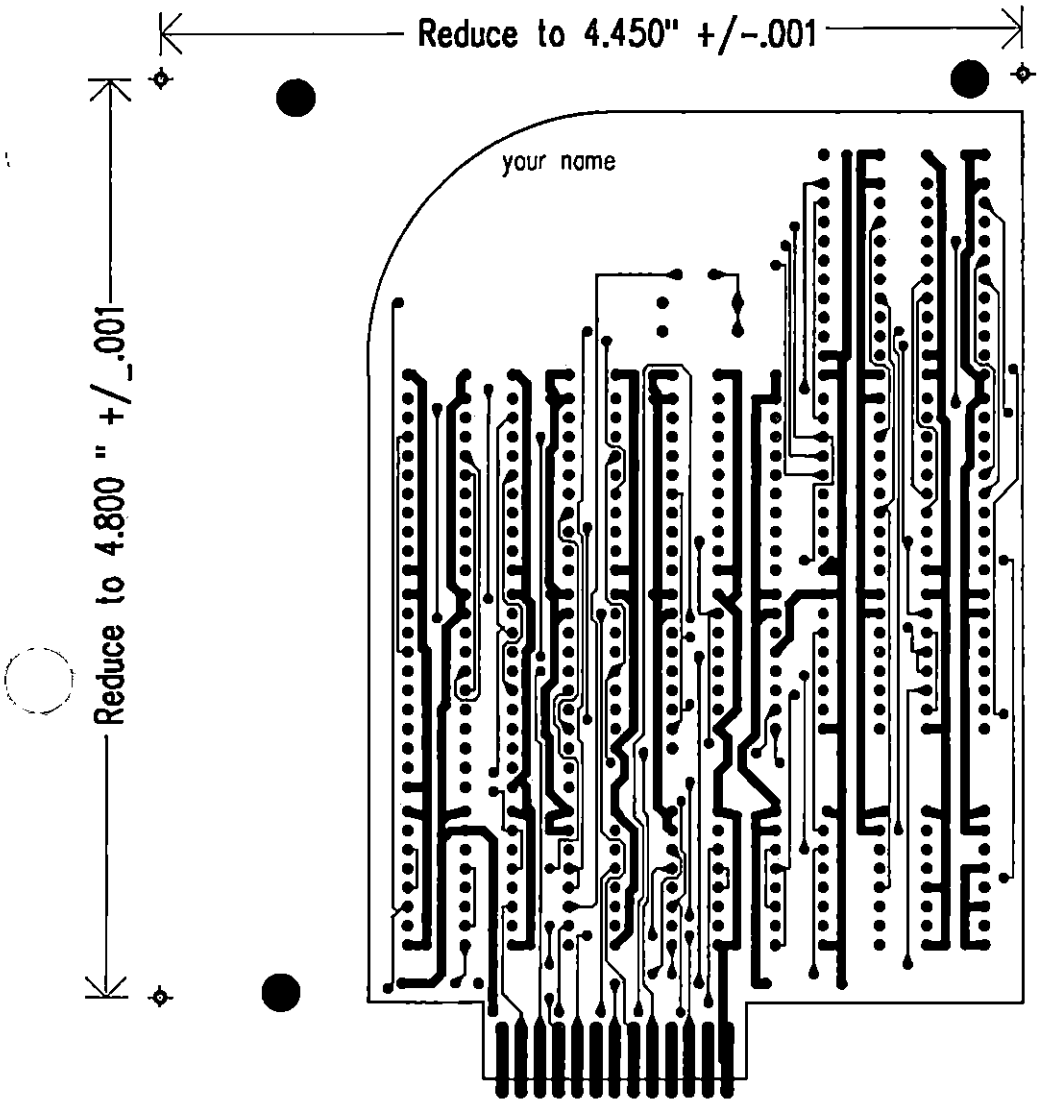



Figure 2.7b

*Figure 2.7b - FINISHED TRAINING BOARD, LAYER 2*

## Chapter 3

# TUTORIAL 2 - CREATING ITEMS AND PARTS PLACEMENT

In this Tutorial we will continue our self-training and practice with two of the major options. These are the Create Routine and the Place Routine.

## 3.1 THE CREATE ROUTINE

(60-90 minutes)
The CREATE option (F3) permits the creation of the Board Outline, Text, 2-D Draw, Copper and Part Decals. We will now use CREATE to create the Board Outline.

In the process, we shall learn to use Text Board Outline and 2-D Lines; Copper and Part Decal will come in later tutorials.

At this point, take a few minutes out and read Volume II, Reference Manual Chapters 5, Section 5.1, 5.2, 5.3, 5.4 & 5.6.

BOARD OUTLINE
(30 minutes)

Go to Figure 2.5 and study the Mechanical Drawing of the board you will design. You will now create the board outline and store it. Your CRT should be blank. Erase the TPRACTIC.JOB file from Tutorial 1, if it is still on your CRT.

Use CTRL-L and CTRL-I to display the two INFO Windows.)

Set Grid to 25; Type G25 <CR> and check that Grid at the screen changes to 25 in the Information Pop-up Window. (You now have a system resolution of 25 mils or 0.025")

Using SETUP, then ORIGIN, move the System Origin by placing the cursor about 2" down and to the left of the center of the screen. Select ORIGIN (F9). The System Reference (0,0) will now be moved to the cursor location, and all location data is now referenced to the new Origin location.

NOTE: The location you have chosen for the new System Origin used as the Board Origin, together with the Board Dimensions, should result in the center of the Board Outline, when created, being approximately at the center of the screen.

Using Box Window (Delete Key), create and enlarge a box window that includes the Board Origin and the area where the board will be created. Alternatively, use PGUP and enlarge the area. (This enlarged area will be the "drafting table" area on which you create the board.)

To create the Board Outline, you will need the Location Window displayed. Use CTRL-L to display this window, if not already displayed.

Select the CREATE routine; then select the BOARD option (F2), then NEW POLY (F1). Get familiar with the Add Corner, Delete Corner, Angle and Complete. Pay particular attention to COMPLETE. Notice that COMPLETE automatically finishes the polygon back at the Begin point. END finishes at the current location of the cursor.

When you feel comfortable, start the board outline. Put the cursor over the Board Origin (cursor position indicator reads X=0 Y=0).

Select NEW POLY. This starts the Board Outline line. Using the X and Y indicators as guides, move the cursor to the right and proceed around counter-clockwise, creating the Board Outline as per the Board Mechanical drawing of Figure 2.5. You will find using the DX and DY values on the Location Pop-Up Window most helpful. This data indicates the distance from the last corner. Make the curved segment of the board a straight 45° segment. The arc will be created later.

Use the Cursor Position indicator to accurately locate all corners. After creating the top segment, including the 45° Angled corner, use COMPLETE and the Board Outline will be closed on itself.

To create the Arc in the top left corner of the board, select MODIFY (F5), place the cursor over the 45° segment, then MOVE ARC (F5) and move the arc to the final location and finish with COMPLETE (F1).

Note that after the Board Outline has been completed, Numeric key 7, RESET, resets the screen display to a scale that contains the entire Board Outline. Exit the CREATE BOARD menu.

TEXT
(15 minutes)

Item 4, Figure 2.6, Design Criteria, asks you to place your name in .150" high letters at the top of the board, on Layer 1.

Be on Layer 1 (use Modeless command of L1 <CR>) and check the level in the Information Pop-up Window. Select TEXT (F1), CREATE (F1), and in response to:

**NEW TEXT STRING**

enter your name <CR>. Make sure your text is on Layer One only.

Note that your name is attached to the cursor and can be moved about. Note the text is 100 mils high with a line width of 10 mils. Select SIZE (F9), and enter:

**150 10 <CR>**

to change the text size to .150". Position the text at the top of the board and use COMPLETE to finish positioning. Notice that while the text is attached to the cursor, it is somewhat difficult to read until fixed in place.

Practice creating text of different size, on different levels, rotating text and deleting text, until you feel at ease with the CREATE TEXT subroutine.

NOTE: If the REAL WIDTH display of SETUP PARAMETERS is set to the text line width (10), the text will be shown at the True width.

CREATING 2-D LINES
(15 minutes)

The 2-D line feature provides a 2-D documentation capability that can be used to generate more complete documentation for your board. We will start your usage of 2-D lines by creating Reduction Instructions and associated Reduction Aiming Points that will be used to reduce your 2:1 drawing to 1:1 artwork (In these training sequences, we will plot the board at 2:1 ready for this reduction.)

Reduction Instructions serve the purpose of creating an accurately referenced dimension in both X and Y for use during the photo reduction sequence. (Look ahead to Figure 3.1 for the completed Reduction Instructions you are to create.)

NOTE: In real life, if you are plotting at 1:1 on a photoplotter you do not need Reduction Instructions.

A Reduction Aiming Marker with the name RAM has been stored in the subdirectory LIB. You will recall it, copy it twice and place the three markers at three different corners on the board and then enter the reduction instructions text and dimension lines associated with the aiming markers as shown in Figure 3.1.

Insure you are on Layer or Level 1. Select 2-D LINES. Select ADD. Enter the name RAM <CR> (This command will retrieve the RAM marker from the LIB directory and display it at the screen). The Aiming Marker will be attached to the cursor. Move the marker up and to the left of the upper left corner of the board and place it, using COMPLETE.

Place the cursor over the aiming marker and using COPY, make a second Aiming Marker and locate it exactly 4.450" to the right of the first Aiming point. Magnify the area where the second aiming point is to be located using Box Window, if necessary. Place the second Aiming Point.

Using ADD, call up the 2-D Line item called Arr200 (a dimensioning arrow) rotate and place it at the first aiming point. Using COPY create a second arrow, rotate it and place it at the second aiming point. Draw 2-D Lines to finish dimensions, if necessary. Exit the 2D LINES menu,

and enter the TEXT menu. Now enter the following text between the arrows, in letters 200 mils high:

**REDUCE TO 4.450" +/-0.001**

This establishes the reducing dimension in the X direction. Using the above sequence, add and locate the third aiming marker, exactly 4.8" below the first marker. Add the dimensioning arrows and reduction instructions:

**REDUCE TO 4.800" +/- 0.001**

Your screen presentation should now contain the board, your name, reduction instructions, and the Aiming Markers as shown in Figure 3.1.

Move to Level 2 and repeat for Layer 2.

Let's now save your Board Outline to hard disk.

There are two methods for saving a Board Outline:

(1) SAVE in the CREATE BOARD subroutine, or
(2) JOB OUT, of the IN/OUT routine.

If there were no text or 2-D Lines on the Board Outline, SAVE in the CREATE BOARD subroutine would be the method used to store the board outline. SAVE does not store text and 2-D Lines however, therefore we will use JOB OUT in the IN/OUT routine.

Select IN/OUT, then JOB OUT. Save the board using, as its name, your initials followed by 1, i.e. XXX1, to indicate the first file in a series of files that will be made during training. Enter the job in your Job Save Log you started in Tutorial One, with the description "Board Outline with Reduction Details and Text".

NOTE: Take care to make these files, name and record in the Job Save Log in the sequence suggested, and save on your hard disk. You will need them from time-to-time for use in the advanced Tutorials. More importantly, you are developing a habit of "SAVING" your work so that when you are doing real boards, you will not lose your database.

Practice with Board Outline, Text and 2-D Lines until you feel comfortable with the subroutines.

## 3.2 CREATING A BOARD DATABASE

(120-150 minutes)
We will now load the part and connection files into memory along with the Board Outline after which we will SETUP the database.

Using IN/OUT, display the XXX1 file, (the Board Outline) at the screen.

We will now load the parts into memory and combine them together with the board. A Parts List of the training board called TPART.ASC has been prepared for your use and stored in the subdirectory FILES. It has been prepared in ASCII using the ASCII Editor contained in the software. (If you want to see the contents, use the command ALT 9. Simultaneously, press ALT and 9 and respond with the Name TPART <CR>)

Select the IN/OUT routine; then ASCII IN, in response to the request for NAME, enter the name of the parts ASCII file TPART and the file is brought from hard disk and read into memory (approximately 3 minutes). Always use the printer to print any errors in the file.

As the file is read into memory the Part Types and Decals are retrieved from the Library on the hard disk, the word WORKING is displayed in the Information window and the file is checked for errors. If errors are found, they are printed out on the printer and must be corrected before being accepted into the board database.

Note at the screen, the parts are all located at the Board Origin, except the Edge Connector (P1), which has been located by including it's X and Y origin location in the Parts List.

(At this time don't concern yourself as to how the Parts and Connection Lists were prepared; a later Tutorial will instruct you as to how to prepare both the Parts and Connection List from a manually prepared schematic, or how to edit a schematic input file from a CAE system.)

We will now enter the Connections file into the database and combine it with the Board Outline and Parts.

NOTE: The Parts List MUST always be read in before the Connection List is entered, otherwise, errors will result since there are no Part pads/pins for the connections to be connected, to present in the database.

A Connection list from the training board schematic has been created and is stored in the subdirectory FILES, under the name TCONN.ASC. This file may be examined using ALT 9 and the name TCONN. Using ASCII IN, read this file into memory where it is merged with the Board and Parts already in the database. The Connection List is checked for errors as it is merged with the Parts. Any errors must be corrected.

After the Board Outline, Parts List and Connection List are merged, create a file using JOB OUT called XXX2 (again, use your initials for the X's), and log it in your Job Save Log with the description "Board Outline, Parts and Connections". Be sure to save and log all your files! You will use them in the Advanced Tutorials.

SETUP

(10 minutes)
With the Board Outline, Parts and Connections loaded into memory, you will practice what you learned in Tutorial 1 on SETUP to set up the database for the board you will design.

"SETTING UP" A NEW BOARD
When starting a new board, it is a good procedure to check, and change if necessary, all the default parameters of SETUP, as they apply to the board. Once these parameters are established for a board, they stay with the file, and do not need to be reset, each time a board is recalled from memory.

Accordingly, let's set up our practice board by going through SETUP, DISPLAY and checking for the following:

1. DISPLAY
Set ALL items on Layer One to Yellow, except ERRORS. Set ERRORS Layer 1 to Red.
Set TOP (Layer 1) to Yellow.

Set ALL items on Layer Two to Red, except ERRORS. Set ERRORS Layer 2 to Yellow.
Set BOTTOM to Red.
Set Connections to White.
Set Background to your choice (Black or Blue)

2. PADS
First, get a printer listing of all Decal Pad sizes by selecting LIST ALL (F1).

Examine the listing. Note the entry for Decal DIP14 as follows:

Pad Stacks for Part Decal : DIP14
Used by Part Types : 7404 7402 7474 7432
7440 7420 7400

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP (1) | 60 | R | 37 |
| BOTTOM (2) | 60 | R | |
| PIN 1 | | | |
| TOP (1) | 60 | S | 37 |
| BOTTOM (2) | 60 | R | |

The above is read: DIP14 has all pads on Layer One as 60 Mil Round with a 37 Mil Drill Hole, except Pin 1, which is 60 Mil Square with a 37 mil Drill Hole. All pads on Layer Two are 60 mil Round.

NOTE: The above pad stack setting is Automatic as the Default setting for all Decal pads as the Decals are created. The Decals are stored in the PADS Library with this pad stack size. If this size is not correct for your design, you must reset in the board database. Once set, this Pad Stack Size will remain with the database.

Let's get some practice changing Pad Stacks. Let's make all pins on Layer 1 75 mil Square.

In response to:

**NAME OF PART DECAL>**

respond:

**DIP14 <CR>**

The Pad Stack data for DIP14 is displayed at the CRT.

Change the Lines:

```
   ALL   T     60    R     37
and
   1     T     60    S     37
```

to read:

```
         T     75    S     37
and
```

COMPLETE.

Make a printer listing and confirm that ALL DIP14 pads on Layer 1 are 75 mil Square.

Additionally, confirm visually by expanding the CRT around the parts and confirm that all DIP14 Layer 1 pads are 75 mil Square.

Now reset to the original setting of 60 mil Round with Pin 1 60 mil Squares before proceeding further.

3. SIZES
Set Board Outline to 12
Set 2-D Lines to 10
Set Part Outlines to 12
Set Tracks to 12 50
Set REF DES to 125H/10W
Set Text to 150H/10W

4. PARAMETERS
Current Level: Off
Net Highlight: On
Tear Drop Pads: On
Copper Hatch: (Leave at default)
Dot Grid: 1000
Real Width: 12
Thermal Line Width: (Leave at default)
Copper Grid: 50

Copper Pour Clearance: 15
Pad/Pad Clearance: 13
Pad/Track Clearance: 13
Track/Track Clearance: 13

5. NET ATTRIBUTES

Using ADD ITEM, Add +5V and set Display to OFF
Using ADD ITEM, Add GND and set to OFF

NOTE: The above, will result in all connections being displayed except +5V and GND. During the placement effort, this will speed up the placement, after which +5V and GND will be turned on at the proper time for routing.

Let's now save the board database that you have created and "Setup" using IN/OUT, Job Out. Save the file as XXX3..

## 3.3 PRACTICING THE INTERACTIVE PLACE OPTION

Please read Chapter 6, PLACEMENT, Volume II Reference Manual before proceeding with the training session on Placement of Parts.

(90 minutes)
Components will be placed in their proper location on the board using the option of PLACE.

We have created a file of the training board to familiarize you with PARTS PLACEMENT before you actually place the parts of your training board, which you saved above. This familiarization file is called TPLACE1. JOB. Call it up now, using the IN/OUT routine, and JOB IN.

TPLACE1 will be displayed at the screen.

Select the PLACE option and study the commands of PLACE. You will notice that there are nine Job routines in PLACE.

Let's now get acquainted with some of the PLACE options using TPLACE1.

MOVE

Select Move (F1). Notice that the menu changes inviting you to "select" the component to be moved. Let's use an IC with the Reference Designation of U1 as the component to familiarize ourselves with the PLACE commands.

In response to MOVE, SELECT, type in:

**U1 <CR>**

Notice that U1 is attached to the cursor at Pin 1. (The latter is because the Part Origin has been made coincident with Pin 1 when the Part Decal for U1 was constructed.) Also, notice that when moving a component, the component is identified in the Information window, if both parts of the window are displayed. This identification includes the Reference Designation, Part Type, Technology and the Part Decal Name. Now move the cursor and notice the part U1 moves about and the connections attached to U1, dynamically rubber band and follow the movement of U1. You can move and place U1 anywhere on the board to the resolution as indicated in the Grid (upper left hand window in screen). Currently, the grid is set at 25 which means you can move the part to the nearest 25 mils.

Suppose you wished to have the components always located on a 50 mil grid. How could you insure that this would happen? Here's how: Type:

**G50<CR>**

Notice that the Grid is changed to 50 mils. Any components placed now will be to the nearest 50 mils. In general, for Autorouting purposes, always use a Placement Grid of either 50 or 100 if at all possible.

Also notice that the PLACE, MOVE command has options that allow you to rotate the component, swap it to the opposite side or layer, run a length minimization, and use an alternate decal while moving the Part.

ABORT is a handy command to use when you have picked up the wrong component. Let's try it. Pick up and move a component. Now, Select ABORT (F9), and notice that the component flies back to its

original location exactly--you don't have to move it back! This is a good feature for trying a new location for a part.

Here is another way of moving and placing a component. Be in PLACE, select MOVE (F1). Now, rather than typing in the Reference Designation, place the cursor over a part (let's use U1 again) and select it by Button #1 of the mouse. U1 is attached to the cursor ready to be moved. You can now move U1 in the same fashion as when you selected U1 by typing in the Reference Designation. Release U1 at any location by using COMPLETE (F1).

Here is a third and final way of selecting and moving components. This is called Global Move, wherein you type in the Reference Designation of the family of components and the system picks up the first numbered member of that family. You move the component to a location and release it with COMPLETE. Then the system picks up the next numbered member of the family. Let's try it. Select PLACE. Select F1, MOVE, and now type in U* <CR>, and notice that the cursor picks up U1. Move U1 to where you wish it to be and release it. Notice that without further action on your part, the cursor picks up U2, you move it to where you wish U2 to be located and release it, and the cursor then selects U3. This action is repeated until all members of the U family have been selected. Try this feature now. To abort this feature, select ABORT (F9); to skip a Reference Designator, select EXIT (F10).

MATRIX MOVE

Using MOVE, as in the preceding, permits a part to be moved to a location as determined by the System Grid, i.e., 50 mils, 100 mils, etc. Quite often, especially with Integrated Circuit parts, it is better to move a part to a predefined matrix. MATRIX (F8) permits the moving of a part to such a matrix. Let's try it.

We have created a matrix of Rows and Columns on this practice board for you. Let's look at the matrix. Select AUTO (F8), MATRIX (F1) and observe the white matrix at the CRT. The intersection of a Row and a Column creates a "matrix" point, on which the Pin 1 of a Part is attached when in MATRIX MOVE. Let's do it. EXIT and go to PLACE (F4), then MAT MOVE (F2). Pick up an IC and move it. Notice that (1) you may not move the IC except to one of the matrix points, and (2) you may not move the part outside the Board Outline. (There are no matrix points outside the board outline.)

You will find MATRIX MOVE very helpful in moving ICs.

ROTATE

NOTE: If at any time you have moved the parts so much that the presentation is confusing, simply call in the board again using IN/OUT and the name TPLACE1.

Let's now get acquainted with Rotate. Rotate is a command that allows the designer to rotate the component 90 degrees Counterclockwise (CCW) about the Part Decal origin (ROT 90), set the component at any predetermined angle or spin the component about its origin (ROT ANY). Each time Rotate is selected, the Part is rotated 90° CCW from its current orientation. Four selections of ROT-90 will result in a 360° rotation.

SET ANG provides the prompt to enter the angle, in degrees to the nearest 0.10 degree, then SELECT the item to be rotated.

SPIN requires you to select the component and spin or rotate it to the desired location and complete.

Select ROTATE (F3), then ROT-90 (F1). In response to SELECT (F1), either type in U1<CR> or place the cursor over U1 and SELECT (F1) and notice that U1 becomes animated and rotates counterclockwise 90°. Initiate Rotate three more times with U1 and notice that it is back at its original orientation.

ALL IC's can be rotated with the Global command of U*<CR>. Try it. Now try U*<CR> three more times.

Next select C1 and rotate it to 52.7 degrees, using SET ANG (F2).

Next select C2 and use SPIN (F3) and rotate it to any angle.

OPPOSITE

Let's now familiarize ourselves with the subroutine of Opposite. Opposite is used to swap a component from the side where it is currently located (normally the Component side or Layer 1), to the

opposite side or solder side (in the Training board, Layer 2). The primary use of OPPOSITE is in the design of boards with Surface Mounted components (SMD), wherein components may be located on both sides of the board.

In these training exercises you will be placing components on both the Top and Bottom layers.

Select OPPOSITE (F4) and in response to the invitation to select a component, place the cursor over U12, or type in U12<CR>. Notice that the component is mirrored about the X axis and moves to Layer 2 after COMPLETE (F1). The color of its outline corresponds to the color selected for components on layer 2 in the Display subroutine of SETUP. Notice that after the component has been swapped to the opposite layer, it can be moved, rotated, etc., as if it were on the original layer.

Using OPPOSITE, select U12 and notice it is mirrored and the component outline color reflects it has been moved to Layer 2. EXIT and the part will remain placed on the opposite layer.

NOTE: When a part is MOVED, a number of functions can be performed with the part. The functions include: ROTATE (F2), OPPOSITE (F3), LENGTH MINIMIZATION (F4), ALTERNATE (F5), SET ANG (F6), SPIN (F7), MATRIX (F8), ABORT (F9). LEN MIN (F4) and ALTERNATE (F5), those not previously described, will be described below.

LENGTH MIN ON THE FLY

You can minimize the length of the rubberband connection going to a part, while moving the part. This will assist you in making the decision to place a part at a particular location.

To use, select a component to move, U12, move the part to a new location, then select LENGTH MIN (F4). The connections to the part will disappear while the length is being recalculated, then the new connection pattern will be displayed. This is an extremely powerful tool for use in deciding the proper location of a part. Use it freely when manually placing parts.

ALTERNATE

ALTERNATE is a subroutine that allows the substitution of Alternate Part Decals for the primary Part Decal for a Part Type. One use of ALTERNATE is Analog Boards, where different decals for discrete parts may be necessary, such as standing a resistor "on-end" rather than flat. Another use is to switch between Dual-In-Line and SMD packaging, for those Integrated Circuit parts that use multi-packaging.

Select R1 to MOVE, when selected, then select ALTERNATE. R1 Decal will change to a decal that has the resistor standing "on-end". Select ALTERNATE a second time and R1 Decal will change to a flat decal constructed at an angle of 45 degrees. Select COMPLETE (F1) to final locate. Repeat with U12 and an SMD Alternate for U12 will appear.

By using ROTATE and ALTERNATE, you are able to obtain various orientations of discrete parts, and thus pack more parts on an Analog board.

Note that on the TPRACTIC board, only the Part Decal for R1 and R2 and the Memory IC's have been created with Alternate Decals.

NOTE: For ALTERNATE to function, the Part Type in the Library must be constructed with "Alternate Decals."

GLUE AND UNGLUE

GLUE and UNGLUE are subroutines that permit a component to be made moveable or immovable, depending upon the selection of Glue or Unglue. Let's say that we wanted to make a component immovable at a particular location, such that both Interactive Placement and Automatic Placement subroutines are disabled For this situation, we would use Glue. Let's illustrate. Place the cursor over U1 or type U1<CR>, the component to be GLUED, and press SELECT (F1) or mouse button #1. Notice that the U1 component becomes animated, indicating the Gluing action. Now try to move the component, using the MOVE command. Note that you cannot move it and you get the message:

**COMPONENT X IS GLUED DOWN**

GLOBAL GLUE/UNGLUE

Let's say that we didn't want any the IC's to be moved either Interactively or Automatically. Here is how you would glue down ALL the components of a family. We would select Glue and then we would type U* <CR>. Notice that each IC becomes animated in sequence as it is "glued down". Upon the completion of Glue, one cannot move an IC until it is unglued.

The Unglue subroutine functions in a manner identical to Glue by the selection of UNGLUE (F6).

Get familiar with GLUE and UNGLUE, since both options are used to control auto placement.

NOTE: Automatic Gate and Pin Swap can occur on Unglued parts only. Use GLUE or UNGLUE to select the parts to insure Gate and Pin Swapping occur or NOT occur, as the case may be.

MOVE NAME

The Move Name routine is used to move the Reference Designation and Part Type names from their Default position underneath the Part Decal to a location outside the Part Decal such that the names can be seen when components are installed on the board. To use Move Name, place the cursor over the name of the component to be moved, and select MOVE NAME (F7). The name and the part type will be attached to the cursor and can be moved and rotated to the desired place. Note that the name can be rotated 90 degrees, a set angle, or spun to any angle. In addition, the display can be Windowed, Expanded and Contracted. You may want to set the Grid to 1 (G1) for finer control when moving the Name. Finish the MOVE NAME using COMPLETE (F1).

Notice at this point we have not discussed AUTO (F8) (AUTO PLACE). These will be addressed at a later time.

## 3.4. PRACTICING THE AUTO PLACE OPTION

Let's now get familiar with the AUTO PLACE Routine and how it is used in the placement of parts.

We have prepared a file for your use in practicing the AUTO PLACE Routine. This file is called TAPLACE1.JOB. Call it up now using JOB IN and display it at the CRT. Note that with the exception of P1 and U21, the parts are unplaced.

In that which follows, you will:

(1) Use the AUTO PLACE feature to bring the parts up on the board, and place them automatically.

(2) Use the GOODNESS OF PLACEMENT evaluators to evaluate the placement.

(3) Automatically Rename the components.

NOTE: AT THIS TIME, IN ORDER TO MORE FULLY APPRECIATE THIS TRAINING EXERCISE IN AUTO PLACE, YOU SHOULD GO TO THE REFERENCE MANUAL, VOLUME II, AND CAREFULLY READ SECTION 6.8. YOU SHOULD NOT, REPEAT NOT, ATTEMPT THIS EXERCISE, WITHOUT HAVING READ THE VOLUME II, SECTION 6.8 MATERIAL.

With the TAPLACE1 file displayed at the CRT, you will notice the following:

(1) The two mounting holes have already been placed in their final location and glued down.

(2) The connector P1 has been placed in its final location, and glued down.

(3) Memory IC U21 has been placed on a matrix point and glued down.

The reason for the latter is there are a number of Memory IC's on the board and the designer desires they be located at the top of the board. We have interactively placed one of the memory IC's (U21) on a matrix point in the area where the memory array is to be located. This component will cause the other memory IC's to be pulled up on the top of the board on the memory array matrix points during Automatic Placement. This "pulling" feature can be used to a great advantage in your day-to-day work in the future, when you desire to force the Auto Place routines to place components in a specific area on the board.

Other preparatory things that have been done to the AUTO PLACE file are as follows:

(1) The Power and Ground connections have been turned OFF. (This should ALWAYS be done with Automatic Placement.)

(2) Two different matrices have been set up: the first is for IC Placement and the second is for Decoupling Cap Placement. If you wish to see these two matrices, be in PLACE, select AUTO PLACE (F8), then MATRIX and you will see the first matrix. This is to be used for IC's. Next, select SELECT (F1) again and you will notice the second matrix. This is to be used for Automatic Placement of the Decoupling Caps. When you exit from MATRIX, be sure to unselect the Decoupling Cap matrix, using UNSELECT (F2), so that the only matrix that has been selected is the IC matrix. If you do not do this and leave both matrices selected, the IC's will be automatically placed on both matrices--a very undesirable situation!

We are ready now to proceed with the Automatic Placement of components on the board.

You will control which components are to be acted upon by the Auto Placement Routines with the use of the GLUE and UNGLUE subroutines. Any component that is glued down CANNOT be acted upon by Automatic Placement; any component that is unglued, CAN be acted upon by Automatic Placement. It is a good habit when you are starting to Auto Place, to glue down all components on the board. Let's do it now.

Select GLUE (F5) and then SELECT (F1) and respond * <CR>.

The *, as you have learned previously, is the wild card command indicating "ALL" components. Note that after you have selected "All" components, all the components articulate as they are Glued down.

We now proceed with the placement of the IC's. To do this, we must now Unglue only the IC's. Select UNGLUE (F6) and then in response to SELECT (F1), enter U* <CR>. Note that ALL IC's are now Unglued, including the Memory chip U21 that is already on the board. We want to keep U21 in its fixed location in order to pull up the other Memory Chip; therefore, we need to Glue it down again. Select GLUE

(F5); then in response to SELECT (F1), use the mouse cursor to select U21, such that it is glued down.

Let's review the status of components at this time. All components except IC's should be glued down; but within the IC's, U21 should be glued down. In short, the only unglued components on the board at this time should be the IC's less U21. Additionally, the matrix that has been selected is the matrix that is to be used for IC's.

It is always a good habit before proceeding into the Automatic Placement Routine, to check the assumptions that you have made. Check to see if the components are Glued or Unglued, using MOVE (F1), as you intend them to be, and check to see if the matrix is the selected matrix you want to use.

Finally, it is always a good idea to make a "Job Save" at this point, such that if you make an error, and wish to abandon, and restart the Automatic Placement sequence, you can always come back to the conditions at which you started Automatic Placement.

## 3.5 PARTS PLACEMENT ON THE TRAINING BOARD

(80 - 110 minutes)
Now that you are familiar with the PLACE routine, including AUTO PLACE, you will now place the components on the training board. To do this, let's call up file XXX3 that you made after you had merged the Board Outline, Parts and Connections and Set Up the database earlier in this Tutorial. Select IN/OUT (F1). Select JOB IN (F1), and in response to the message:

**JOB INPUT FILE NAME**

type:

**XXX3 <CR>**

the file you created previously. File XXX3.JOB is displayed and we are ready to start the placement of parts.

The first category of parts to be placed on either the training board or your real boards are the Fixed Location Parts. "Fixed" components are those components, such as the connectors, Mounting Holes, etc.,

whose locations are not variable but rather are determined usually by the board Mechanical drawing or Design Criteria. On the training board, there are a number of fixed components. These are:

1. P1, the Edge Finger Connector
2. M1, M2, M3, Moire Pads

CONNECTOR PLACEMENT

Let's start with the Edge Finger Connector, P1. Surprise! We have placed P1 for you, using the Parts List. Go to Figure 2.4 and locate part P1. Note the 700 -300 following the part description. This is a command to locate the part 0.700" in X and -0.300" in Y from the Board Origin. This, incidentally is a good practice for you to follow in the future on all "Fixed" parts where possible, especially Connectors. Use the PARTS LIST to place Fixed parts! Had we not used the Parts List to locate P1, it would have been located at 0.0 and you would have placed P1 as follows:

You would use the PLACE Routine and move P1 from the 0,0 position and place it to a ±1 mil accuracy as indicated in the Design Criteria of Figure 2.6.

Select MOVE (F1), then SELECT and type in P1, and the Edge Connector would be attached to the cursor by its Pin 1. Move the component to the location as indicated in the Design Criteria. Note that the Edge Finger is to be 0.700" from the Board Origin in X and -0.300" in Y. Notice that it may be difficult to place a component, using the cursor, to the nearest .001". What do we do? The answer is that we would use a Search command that allows us to move to the nearest .001", as follows:

Insure that the grid is set at 1 mil, by typing:

**G1 <CR>**

Type:

**S700 -300 <CR>**

and notice that the cursor moves to this position, to the nearest .001". Be careful not to move the cursor, and final place the component at this location using the COMPLETE (F1) key.

This S command, standing for Search, followed by the X coordinate, a space, then the Y coordinate, is an excellent way of locating components or items to the nearest .001" (Grid must be set to .001") and will stand you in good stead at various times during the design of a board.

Caution: Take care NOT to reset the Origin while using a Grid of One. To do so will shift the routing grid off the desired routing grid. If it is necessary to shift the Origin while on a Grid of One, take care to reset the Origin and realign the routing grid before routing (See Section 4.5.6)

HINT: This S command is one of a number of Modeless commands that are available and are great time savers. Go to Section 1.4, Chapter 1, Volume II, Reference Manual for a complete listing of Modeless Commands.

After final locating a part that is to be fixed in a location, it is a good idea to glue it down to prevent its inadvertent movement. Select GLUE (F5), and then select the part P1 and observe it being animated. Then select MOVE (F1), and test try to move P1. It should NOT be movable.

MOIRE PAD PLACEMENT

There are three Moire pads (M1, M2 and M3) on the board that will be used for front to back registration of the Artwork by the Board Manufacturer. These pads are usually located near three different corners of the board so as to provide registration guidance in both X and Y. We will use ON-THE-FLY, rather than ECO to add these MOIRE parts, since their addition does not change the circuit and hence does not necessitate an ECO.

Select IN/OUT (F1), ON-THE-FLY (F8), ADD PART (F5), KEYBOARD (F2) and respond:

**MOIRE <CR>**

to the prompt:

**NAME OF PART TYPE FOR NEW PART**

and "**M**" for the prefix.

Use SELECT (F1) to create M1, M2 and M3 and place them just outside the Board Outline at the lower left corner, the upper left corner and the upper right corner. After Placement, glue them down. This completes the Moire Pads Placement.

NOTE: If you are planning on using a Gerber photoplotter, use Moires for Registration, rather than the Reduction Aiming Markers discussed for wet ink or matrix plotters. In the training board, we use both Reduction Markers and Moire pads for the purpose of illustrating both 2:1 ink plots and photoplotted 1:1 artwork registrations.

UNFIXED (MOVEABLE) COMPONENTS

We are now ready to place those components without a specific location and where the designer makes the decision as to whether one location is preferred over another.

We see from the Design Criteria that the IC's are to be located Vertically with Pin 1 on the top left corner. We see at the CRT the IC's are Horizontal. Therefore, we will rotate them to the Vertical position. Select PLACE, then ROTATE (F3), then ROT-90 (F1). In response to SELECT, use the Global command of U* <CR> to rotate all the ICs with one command. Notice the ICs are now Vertical. Unfortunately Pin One is in the lower right corner. You will, therefore, need to rotate twice more, using the Global U* command until all ICs are Vertical with the pin one in the upper left. Go ahead and do it, then Refresh the CRT with Numeric Key #1.

We could place the ICs manually or using AUTO PLACE. Let's use AUTO PLACE.

We will need a Placement Matrix for the auto placement of the IC's. The generation of a Placement Matrix is in the subroutine AUTO PLACE, therefore, select AUTO (F8). Notice the options associated with AUTO. For the time being, we will be concerned only with MATRIX (F1), and we will use MATRIX to generate the Placement matrix upon which we place the ICs.

A Placement Matrix is simply a number of rows (horizontal) and columns (vertical) with a spacing adequate for the placement of IC's and a number of placement points (intersection of rows and columns) to at least equal the number of components being placed on the placement

matrix. We have a total of 20 IC's on the schematic. Therefore, we will need at least 20 matrix points. In examining the configuration of the board (see Figure 3.1), it appears that a combination of four horizontal rows and six vertical columns, giving us 24 matrix points, is the best arrangement for this board.

Set Grid = 50 (so the matrix and thus the parts will be on a grid of 50). Use the Modeless command of G50 <CR> To create a Placement matrix, be in AUTO, select MATRIX (F1); then CREATE (F3). And in response to the message:

**NUMBER OF ROWS**

enter:

**4**

In response to the MESSAGE:

**SPACE BETWEEN ROWS**

enter:

**1125**

(900 mils between rows).

In response to:

**NUMBER OF COLUMNS**

enter:

**6**

And in response to:

**SPACE BETWEEN COLUMNS**

enter:

**550**

(500 mils between columns).
Spacing is in mils. Upon completion of entering the Placement Matrix, move the cursor to the center of the CRT and a white matrix will be displayed with its lower left corner attached to the cursor. Using MOVE (F5), move the matrix so that it is located on the board as appropriate

and release with COMPLETE (F1). Notice that the matrix has four rows and six columns with spacing as entered above. Each intersection of a row and column form a point on the Placement matrix where an IC will be located with its origin (Pin 1) coincident with the placement matrix grid point. The location of the matrix on the board should take this into consideration. A location with the lower left matrix point at 200 900 is a good beginning. You can always move the matrix later.

Example: Figure 3.1. Note that the top left four matrix positions are not used. How do we keep the AUTO PLACEMENT routine from placing IC's on these four matrix points? Answer: By deleting the matrix points.

To delete a matrix point, place the cursor over the matrix point and select DEL PNT (F6) and a circle will enclose the matrix point, indicating the point is no longer a part of the matrix and hence will not be used by Auto Placement. Delete all four top left points.

NOTE: When exiting the MATRIX menu, the Matrix will disappear.

We see from item 10 of the Design Criteria, that the Memory IC's are to be located at the top of the board. We can "force" AUTO PLACE to do this if before we AUTO PLACE IC's, we place one of the Memory IC's on a top matrix point. Using MAT MOVE (F2) (Not MOVE), select U20 (a memory chip) and locate it in one of the two top matrix points.

We are ready now to move the Integrated Circuits up on the board and we want the ICs (except U20) unglued and all non-IC components plus U20 glued down so they will not be involved in auto placement. Here's how we do this. Be sure you understand the logic and procedure of what we will be doing, because you will use this to great advantage as you design boards in the future.

Let's glue ALL parts down by selecting GLUE (F5) and then typing * <CR> and observe that all Parts articulate as they are glued down. Test by trying to move a part. You should not be able to move a part. Now unglue only the ICs by selecting UNGLUE and entering U* <CR>, observing that all Us are unglued.

We must glue U20 down on the matrix point in order that it pulls the Memory IC's to the top of the board. Use GLUE and select U20. Check that it is glued.

The situation now on the board should be that all discrete components, Resistors, and Capacitors, and U20 should be glued and not moveable, but all other IC's should be unglued and movable. Check that this condition exists by trying to move various items.

IC PLACEMENT

There are two ways in which you can place IC parts on the board. The first is the ALL Interactive method where you will make the decision as to which matrix point an IC is to be located, based on your study of the flow of the Schematic or the Design Criteria.. The second method is to use AUTO to autoplace the ICs up on the matrix and automatically sort them so as to minimize the total connection length.

We will use the AUTO PLACE method in the following placement training session.

Within the ICs, the type of Net Length Minimization will greatly affect the results of the automatic parts placement, and thus it is essential to select the correct type of LEN MINIMIZATION. Go to PLACE, AUTO and NET ATTRIBUTE (F2). Set ALL to ON and Len Min Type to MIN. Set +5V and GND to Display OFF.

NOTE: Turning a net to Display OFF in NET ATTRibute does three things: (1) the Net is NOT displayed, (2) the Net is NOT Length Minimized, and (3) the Net is NOT routed in the Autorouter. We have turned +5V and GND OFF during Auto Placement since they do not contribute or help the Auto Placement calculation. The reason for this is a PWR and GND connection goes to the same pin on each IC and thus equal weight is given to each IC. The Auto Placement algorithm seeks to "bias" for part when selecting its location.

You have some additional control over the placement of parts so as to bias the parts location against Long connections, Angled connections or Long Angled connections or use a Normal connection distribution. (See Section 6.11.3 of Volume II for a discussion on CONN BIAS.) Use AUTO (F8), CONN BIAS (F3) and set to NORMAL (F1) on the Training Board.

AUTOPLACEMENT

The next effort will be spent in auto placing the ICs.

At this point, make a Job Save as XXX4 and record it as "Board Ready for Placement." Always make a save just before going into Placement. In this manner, if you don't like the results you can return immediately to the "Board Ready to Place" file and start anew. Very important!!

We are now ready to Autoplace the ICs.

Select AUTO (F8), then INIT PLC (F4). INITIAL PLACE is used to Auto locate the IC parts on the matrix in an approximate location. After a short time, the ICs are placed on the matrix. Observe the "Working" prompt messages during the sequence.

Inspect the results of INITIAL PLACE. The memory array should have been placed at the top of the board. Use LEN MIN (F7) to get an indication of total connective length in inches. Record the length. Make a save at this point as XXX5 in the event you wish to return and use the Initial Placement.

OPTIMIZE (F5) is used after Initial Placement to obtain a finer and perhaps better placement.

It may be possible to allow or obtain a shorter connector length and have a better placement using the OPTIMIZE (F5) feature of AUTO. Let's try. Record the correct connection length, then select OPTIMIZE, then SWAP PARTS (F1). You have the option of swapping all parts on the board with other parts either MANUALLY or AUTOMATICALLY. Use AUTO. Select AUTO (F2) and observe the "working" and prompt messages.

Upon completion, use NET LEN to measure the new total net length. You will notice that it may be somewhat less than the net length resulting from the Initial Placement.

When you are satisfied with the placement of ICs, use GLUE and U* <CR> and glue all IC's down. Make a job save XXX6.

DISCRETE PARTS PLACEMENT

The Design Criteria of Figure 2.6 requires Y1, R1 and R2 to be placed at the top of the board. Unglue the parts and use MOVE to locate Y1, R1 and R2 at the top of the board. The Design Criteria also requires the Filter Caps C1 and C2, to be located near P1. Do this now. Glue down Y1, R1, R2, C1 and C2.

DECOUPLING CAPS

You are now ready to place the Decoupling Caps.

You may place them on top of the IC's (1 cap/IC) either with or without a matrix.

Let's use a matrix and AUTO PLACE. You can either create a new matrix or use the IC matrix but move it to the location of the Decoupling Caps. We will use the latter. Use AUTO, MATRIX, and place the cursor over the lower left corner of the matrix and select MOVE (F5). Using the cursor, move the matrix .100 up, such that each matrix point falls above each IC, pin 1.

With the Decoupling matrix in place, we are ready to take the GLUE/UNGLUE action to result in only the Decoupling Caps (C3-C22) being unglued. Proceed as follows:

Glue All: *** <CR>**

Unglue all Caps: **C* <CR>**

Glue: **C1 and C2**

At this point, all parts except the Decoupling Caps are glued. Test for this condition.

NOTE: At any time the results of AUTO PLACE are strange or unexpected, it most likely can be attributed to not having the proper GLUE/UNGLUE setup as required for the action being taken. Test!

Select AUTO PLACE and INITIAL PLACE (F4). If all has been set properly, the Decoupling Caps will be located above the IC's. If not, replace as necessary.

## 3.6 GATE AND PIN SWAP

If at this time you are satisfied with the general placement of parts, and if you are permitted to setup gates and pins on the board, then it is at this point that Gate and Pin Swapping is performed.

We are now ready for Gate and Pin Swap, with the intention of further reducing the connection length and thus improving routability.

Using SHOW PLACE (F8), then NET LEN (F3), obtain and record the X and Y Net Length. This number represents in inches the total X component and Y component of all Nets on the Board.

For Gate and Pin action to occur, two conditions must be filled:

(1) The Part Type in the Library must be created to allow swapping.
(2) The Part Decals of the parts to be swapped must be unglued.

The Part Types of the Parts used in this Training exercise have been created to allow swapping.

You must UNGLUE the Parts. Do this by using UNGLUE and * <CR>.

Select PLACE, AUTO and OPTIMIZE (F5). Then select SWAP GATES (F2).

In response to the prompt:

**GATE/PIN SWAP REPORT FILE NAME**

select <CR> and a listing of swapped gates will be printed. This represents the Engineering Change Order that must be implemented in the originating schematic in order that the originating schematic and board database track each other. If your originating schematic is PADS-LOGIC, respond with a file name. This file name is automatically assigned the extension of .ECO, and can be used to automatically update the PADS-LOGIC schematic.

Let's print out the Swap Report by responding <CR>.

You may either swap gates MANUALLY or AUTOMATICALLY.

First try MANUAL. Use Select and select the pin on a gate to be swapped. If swappable, the associated connections' color will change to indicate the selected gate. Select the other gate and confirm your intention to swap.

If NOT swappable, error messages will result.

Let's swap automatically. Select AUTO (F2). The gates will be swapped where the swap will result in a shorter connection length, and a Swap Gate Report will be created.

Inspect the report as printed.

```
*PADS-ECO*
*SWPGATES*
*REMARK* 7404
U6.A  U6.B
U2.E  U2.F
*REMARK* 7402
U3.A  U3.B
*REMARK* 7432
U5.A  U5.D
```

The report format is read as follows: "U6.A and U6.B were swapped, U2.E and U2.F were swapped," etc.

SWAP PINS. Select AUTO, OPTIMIZE, SWAP PIN (F3) and AUTO (F2).

Examine the resulting printout of swapped pins.

Your board is now placed. Double check the placement, using the Placement Check List at the end of this chapter as well as the Auto Placement evaluators in AUTO and make any adjustments needed. Save as XXX6.JOB. Record as Board Placed.

## 3.7 GOODNESS OF PLACEMENT EVALUATORS

In addition to the CONNECTION LENGTH indicator, AUTO has "Goodness of Placement Evaluation Routines" that you may use to evaluate the placement results. These are located in the AUTO, SHOW PLC (F8) subroutine.

The first Goodness Indicator is HISTOGRAM (F1). With the training board displayed at the CRT, select HISTOGRAM (F1) and respond to the ROUTING GRID size prompt with:

**25 <CR>**

A Horizontal Density Ratio Graph will be displayed across the top of the board, and a Vertical graph will be displayed along the left side. The graphs represent the ratio of "connection-To-Routing Channel Ratio" for each 25 mil routing channel in both the X and Y direction. The "peaks" in the graph represent potential routing problem areas.

The second "Goodness" Evaluator is DENSITY MAP (F2). This map shows the board area, color-coded in accordance with the connection density in various areas of the board, with RED used for potential problem areas. Select DENS(ity) MAP (F2) and respond to the prompt:

**DENSITY MAP GRID WITH 100 <CR>**

In a dense board, you would heed the RED area Hot Spot warnings and continue to refine the placement so as to eliminate the dense connector "Hot Spots."

## 3.8 RENAMING OF PARTS

The Design Criteria (Figure 2.6) requires the IC's to be renamed in sequence from the upper left of the board to the lower right.

This automatic routine is used to renumber Reference Designators by row or by column. This routine is normally used after placement has been finalized. This routine assumes that each Reference Designator is made up of an alphabetic "prefix" such as "U" or "CR" and a numeric "suffix". Only the numeric suffix is altered by this routine.

You will need a Rename Matrix for use in AUTO RENAME. Go to PLACE, AUTO, then SET MATRIX (F1). Use UNSELECT (F2) and deactivate the matrix used for Part Placement. Create a new matrix with a greater number of matrix points. Using CREATE (F3), make a matrix with 10 rows with a spacing of 500 and 12 columns and a spacing of 300. Place the new matrix so as to cover all the components on the entire board.

NOTE: A finer matrix than the placement matrix is required for the proper sequencing of the REF DES. Should you encounter an incorrect sequence, make a Rename matrix with more matrix points.

Upon entering the RENAME (F7) routine, the following prompt will appear:

**RENUMBER HORIZONTALLY, OR VERTICALLY (H/V)? [H]**

If "H" is entered, then the Parts are renumbered in rows, otherwise, they are renumbered in columns. Use H.

The parts may be renumbered starting at either the left or right edge of the board going in ascending order to the opposite edge. This is selected with the following prompt:

**RENUMBER LEFT TO RIGHT (L/R)? [L]**

Use L.

The parts may also be renumbered starting at either the top or bottom edge of the board going in ascending order to the opposite edge. This is selected with the following prompt:

**RENUMBER TOP TO BOTTOM (T/B)? [T]**

Use T.

Having selected the order and direction in which the parts are to be renumbered, the following prompt will appear:

**REF-DES PREFIXES TO BE RENUMBERED: [ALL]**

The parts to be renumbered are selected by specifying the alphabetic prefix. Several prefixes may be listed separated by commas or spaces, i.e., U,R,C,CR,Q. To renumber all parts, enter just an asterisk "*" in response to this prompt. Use U since you are interested in renaming IC's only.

For the ECO Report File, use the same ECO file name as used for Gate and Pin Swapping. This will be used to update the schematic. After the prefixes have been specified, the parts will be renumbered and the board will be redisplayed with the new Reference Designators. The ECO will contain the "was-is" Ref Des. Inspect the results by using ALT 9. Enter the name of the ECO file and the file will be displayed.

The Design Criteria requires the Part Names to be centered under the component and reading right side up. Select PLACE, MOVE NAME (F7) and manipulate the names as required.

Save as XXX7 "Placed, Renamed and Ready to Route."

## 3.9 GROUP OPERATIONS

Group Operations can be used to bring together a number of components, tracks, connections, etc., into a "Group," and manipulate this Group. The group can be moved, rotated, deleted, mirrored, copied, (Step-and-Repeated), and CUT and SAVED on the hard disk for use in another design.

In this Section, we will become familiar with Group Operations. A special job file called TGROUP.JOB has been created for your use in Group Operation. Using IN/OUT, call in the job called TGROUP, and display at the CRT.

You will notice there is a small analog circuit in the lower left-hand side of the board. After TGROUP is displayed at the CRT, Select PLACE, and then GROUP (F9). Use DEF GRP (F1) to define the group, by placing the cursor in the lower left-hand corner of the group, and pulling the cursor until the Analog circuit is enclosed within the rectangle formed therefrom. Then use COMPLETE (F1) to complete the group. Use (F1) to move the Group around. Use (F4) to rotate the Group.

Notice that the Group rotates in 90° increments counterclockwise. Use OPPOSITE (F5) to move the Group to Layer 2.

Move the Group to its original location in the lower left corner of the board. We will now Step-and-Repeat the circuit Three additional times, so that there are Four analog circuits on the board. Using COPY (F2), copy the Group and move it to one of the three vacant places on the board, and COMPLETE. Repeat the operation two times such that there are Four analog circuits on the board. SAVE the file. Note that the system renames the parts on the "copied" circuits, so that duplicate parts do not exit on the board.

You will now see how a Group can be saved and used in another design. Create a Group from one of the analog circuits, and use F6 CUT. Give the Group the name, CUT1, and the group is saved in the FILES directory as CUT1.

We will use this analog circuit on a different board. Bring in the board called TPRACTIC.JOB, using JOB-IN. Select PLACE, then GROUP, then PASTE (F2). In response to the prompt:

**ENTER FILE NAME**

Respond:

**CUT1 <CR>**

and the group called CUT1 will be brought in, and can be placed on the Board called TPRACTIC. (As an additional exercise, use CREATE (F2), then BOARD (F2) , then MODIFY (F5) to modify the Right side of the Board Outline of TPRACTIC to make room for the new group). Notice the REF DES of the Group are integrated automatically into the REF DES sequence of TPRACTIC.

## 3.10 GOOD COMPONENT PLACEMENT GUIDELINES

A good placement of components is a placement that (1) observes all the restrictions placed upon component location and routing of tracks, and

(2) provides the easiest solution in routing of connections. There are a number of general rules or comments relative to placement as follows:

1. A good guideline to follow is to, at the very beginning of placement, place all the components that have a fixed location, that is, those parts where the designer has no choice of location. Amongst these components are such items as Connectors, some IC's that require a specific location, Mounting Holes, Moire pads, etc.

2. Connectors, because of their typically heavy connectivity pattern, need sufficient room between the connector and the first row of IC's to exit the connector. A good rule is to allow at least .5" to .75" or even more space between the connector and the nearest row of IC's.
3. Next place those components that have design criteria restrictions which must be met such as, "this part must be only one-half inch from that part," "this part cannot be located next to that part," "this part must be near the board connector," etc.

4. Following the placement of the above categories, one can now start placing those components over which the designer does have freedom of choice. Whether these are placed using Auto Placement routines or Interactive techniques and using the schematic as a guide, the following general comments are applicable.

a) A Part's pads should fall on the routing grid that will be used during autorouting. This should be 100, 50, 25, 20, 16.67, 12.5 or 10 mils. Set up the system (Grid) before starting the Placement effort.

b) As a general observation, a Placement which minimizes connection length should always be a major and, indeed, a primary consideration. The theory behind this philosophy is that the shorter the connections, the less the copper tracks to be routed. Consequently, on balance, unless there are other mitigating reasons for not so doing, select locations of Parts where the connection length is at a minimum after Length Minimization.

c) While Connection Length Minimization is desirable, one of its undesirable side effects is that in many cases, the use of Minimum Connection Length as the prime Placement criteria, results in "hot spots" or grouping of connections, usually in the center of the board, such that there are more connections going through the area of the hot spot than there are routing channels available. Consequently, always after

minimizing the connection length, study the board using the "Goodness of Placement Evaluators" and if indeed there are critical hot spots or concentration of connections, don't hesitate to make local adjustments to the component locations so as to spread the connections away from the hot spot areas.

d) A large number of long diagonal connections is highly undesirable. Better the flow of the connections be vertical and horizontal rather than diagonal. Having given this advice, one readily admits that it is impossible to eliminate all the diagonals. The point to be made, however, is to try to minimize them, especially the long ones, since the long diagonals will be difficult to route and probably will result in several feedthrus or vias, during the routing effort.
e) The placement of component pads with respect to other component pads is critical in the sense that the pads should be lined up in X and Y such that pads are hidden behind other pads in the same routing channel and do not block routing channels. An example of the worst possible placement is to have two IC's side by side with their pads staggered 50 mils, so that the pads of one IC block the routing channels of the other IC.

f) In general, it is better to have all of the IC's on the board in one orientation, rather than some vertical and some horizontal.

g) PWR and GND connections do not contribute to the decision to place a part at a particular position. In fact, displaying the PWR and GND connections at the screen tends to obscure the placement intelligence obtained from displaying the signal connections only. Moreover, the calculation time of LENGTH MINIMIZATION is greatly increased if Power and Ground connections are included in the calculation. It is important, therefore, that PWR and GND connections be turned OFF in SETUP, DISPLAY, NET ATTRIBUTE.

## 3.11 ADDITIONAL AUTO-PLACEMENT CONSIDERATIONS

Components can be placed on both sides of the board with the AUTO Automatic Placement Routine. For example, Surface-Mounted Devices (SMD), can be simultaneously and automatically placed on both the top and bottom layers of a board. Proceed as follows:

If you know which components are to be on the top side, and which components are to be on the bottom side, the procedure is straightforward. First, create a matrix on the top side, and then create a second matrix; and flip it to the bottom side. Next, move the components that are to be on the bottom side over to the Bottom Layer, using the F4, OPPOSITE Routine. Once these two steps have been taken, proceed with the Autoplacement Routines, as if you were placing on only the top layer. The AUTO II Automatic Placement Routine will automatically arrive at the best placement for both top & bottom layers.

If on the other hand, you do not know in advance which components should be on the top and bottom layer, proceed as follows:

Create a matrix on the top side which has enough matrix points to contain all of the digital components that will be on both layers. Proceed to automatically place all the components on the matrix points on the top layer. After automatic placement, study the results and make a determination as to which component should be on the top and the bottom layers. Flip the components that are to be on the bottom layer to the bottom layer, using the OPPOSITE Routine. Create a new matrix for only the top layer, and a new matrix for the bottom layer; and move it to the bottom layer. Proceed to place on both layers, using the Automatic Placement Routine, AUTO .

In the Example Board used in this Tutorial, the components were standard IC's, with 300 mils between rows of pads, and containing only 14 and 20 pad components. In most instances, boards will be more complex than this training board, and may contain both 300 mil and large 600 mil IC's. What do you do with complex boards such as this, with large components that overlap a number of matrix points?

ANSWER: If you know where you wish the large components to be located prior to Automatic Placement, place these components in this location and glue them down. Then using the "Delete Matrix Point" feature, delete all the matrix points that fall underneath this large component, so that components will not be placed on these matrix points during automatic placement. Proceed to automatically place the components, as discussed previously.

If on the other hand, you do not know where you wish the large complex IC component parts to be placed, create a matrix that will contain ALL the IC parts to be placed. Proceed to place the IC components on this matrix, using Automatic Placement. Upon completion, you will note that there may be parts located on the matrix points that lie underneath the large complex components. At this point, Glue down all the large complex IC's, and then use the interactive MOVE Routine to re-adjust the IC parts in the immediate area of the large complex parts, such that they are no longer underneath the complex part. This will mean undoubtedly, that some of the parts may no longer fall on the placement matrix. However, this should not pose a problem, especially if you insure that the parts are placed on a 100 mil system grid or the routing grid.

What about the situation where you have a mixture of .600" between pads and .300" parts and a large portion of the parts are .600" parts as the standard 14/16/20 .300" pin parts. In this instance, create a matrix that has enough matrix points for all the Digital parts and separate the matrix points with sufficient space to contain the largest IC on the board. This will result in the matrix extending beyond the board outline but that is OK. Proceed with Auto Placement, both INITIAL and SWAP PAIRS. What you have at this point is the correct relative position of each part with respect to all other parts on the board. However, many of the parts are outside the board. You must interactively move the parts onto the board while retaining the relative position of each part. Starting at the lower left hand IC, move it to the lower left corner of the board in its final position using PLACE and MOVE. Repeat with the next IC and place the entire lower row of IC's on the board in their current relative position. Repeat for each row until all parts are on the board. You will probably need to slightly adjust the relative position of some of the parts to make them fit on the board; however, by carefully observing the connection pattern, this adjustment can be made without major disturbance to the relative position pattern.

Next you need to understand the difference in the Discrete Parts Auto Placement algorithm and the Digital Parts Placement algorithm. First, the Discrete Part Auto Place routine is not designed to Autoplace Analog where all parts are Discrete. Instead, it is designed to assist in the placement of the Discrete parts that are included on a Digital board.

When so used, the DISCRETE routine does NOT place the discrete parts on a matrix point as does the Digital Auto Place routine. Instead,it

is used to move the Discrete parts close to the IC to which it is attached; you then final place the Discrete parts. DISCRETE treats the connections between a discrete part and the IC's to which it is connected as "rubber bands", and the DISCRETE part is "pulled" close to this associated Digital parts in direct proportion to the number of connections between the parts. Once this is accomplished, the designer, using MOVE, ROTATE and LEN MIN ON-THE-FLY, positions the discrete part in its proper location with respect to the Digital part.

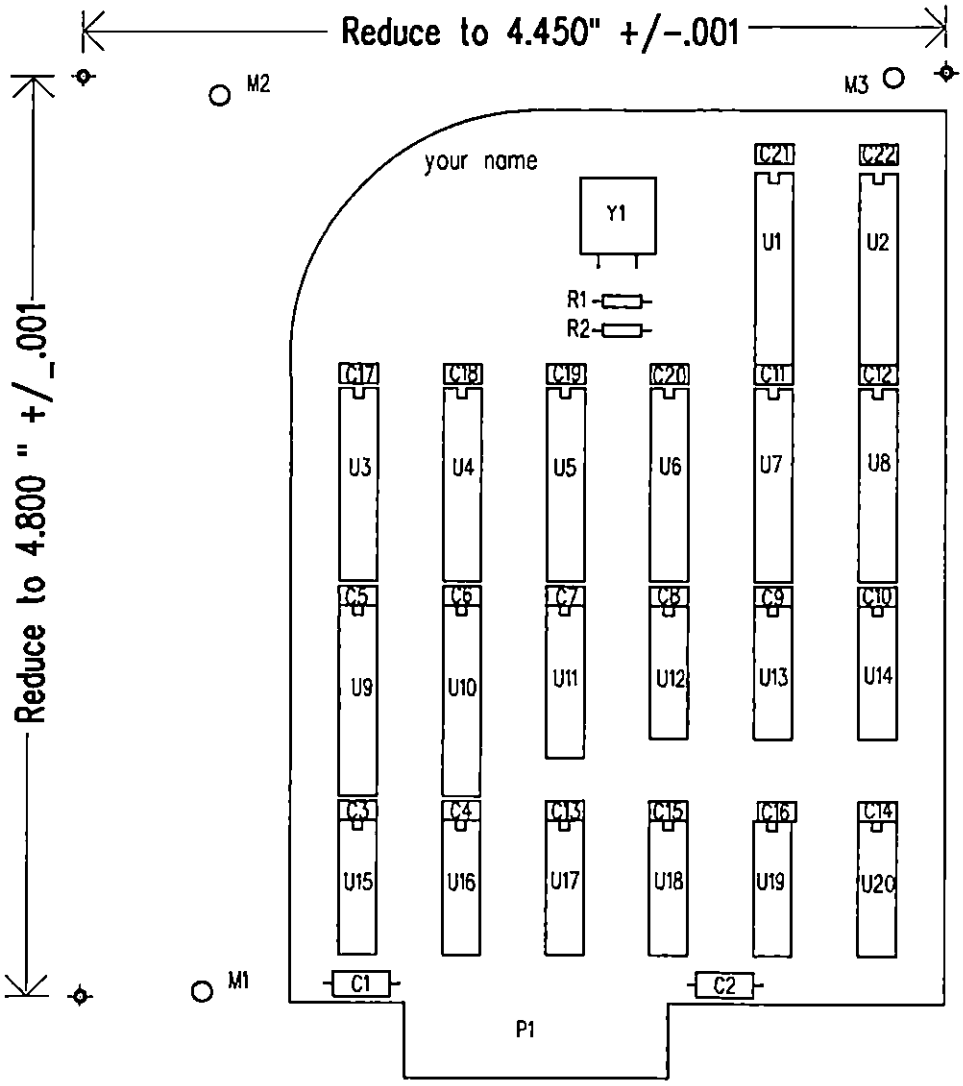



Figure 3.1

*Parts Placed*

## Chapter 4

# TUTORIAL 3 - CONNECTION ROUTING

In this Tutorial we will use the Interactive Router, the Heuristic Routers, the Maze Router and the Shove Router to convert the Power, Ground and Signal connections of the Training board into tracks.

The approach we will use is to, first, acquaint you with how to use the ROUTE Routine and its subroutines. After you have mastered the routine, you will then route the connections on the Training board, starting with Power and Ground, followed by Memory Signal connections and, finally, the remaining Signal connections.

## 4.1 THE ROUTE ROUTINE

Let's get acquainted with ROUTE. Call up and display the special file TROUTE.JOB. This file has several connections on it (see Figure 4.1) and has been prepared for your use in learning the ROUTE Routine. Connections 1, 2, 3 and 4 are 50 mil wide--Power and Ground tracks and 5, 6, and 7 are 12 mil wide Signal connections. All Part Pads except Pin 1 are 60 mil Round and the Air Gap is 13 mils; Pin 1 is 60 mil square.

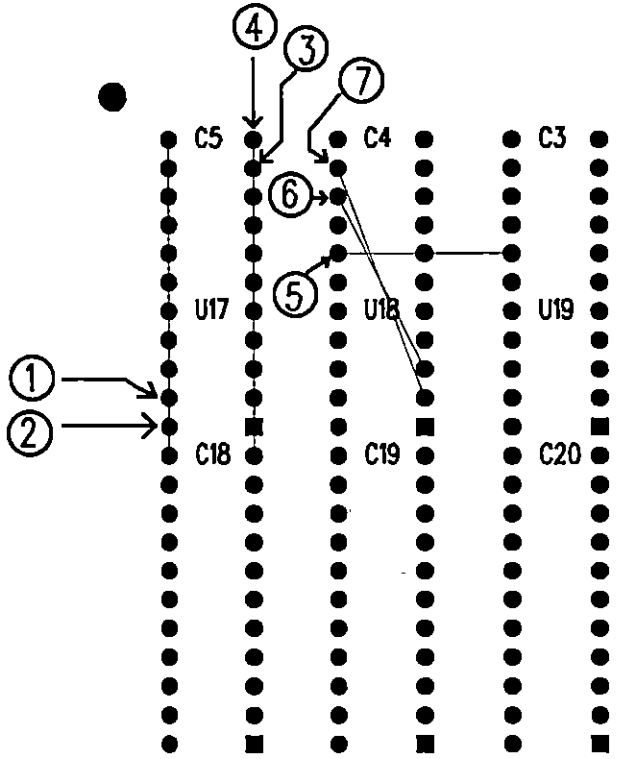



Figure 4.1

*Figure 4.1 - PRACTICE ROUTING (TROUTE.JOB)*

The following are some general guidelines that should be observed during routing.

1. Route horizontal tracks on Layer 1 (3 and 5, etc.) and vertical tracks on Layer 2 (4 and 6). On tracks with significant length segments in both X and Y, use a Via to make the transition from one layer to another so that the X segment is on Layer 1 and the Y segment is on Layer 2.

2. Have Grid set to the routing grid as determined by signal track width plus air gap. (In this practice board, this is 12 + 13 = 25 mils).

3. Have Layer 1 and Layer 2 as two distinctly different colors at the screen. (Check Display now.)

4. Expand Window such that you are working with an area as covered by not more than three to four 14-pin IC's.

5. Route tracks with or without their true width--your choice, but we think True Width is much better.

6. Clean up the tracks as you go, do not leave deliberate errors.

7. Start from one side of the board and progress or sweep across the board--don't jump around from area-to-area on the board.

8. Have Highlight ON.

Let's now practice routing the Power and Ground connections of File TROUTE.JOB (connections 1, 2, 3 and 4).

Have the Grid set to 25 and the Display colors set so as to distinguish between layers. Highlight ON.

Power and Ground tracks are usually routed Vertically between the rows of pins on IC's, on Layer 2.

Have REAL WIDTH set to 12 to show all tracks at or above 12 mils wide with Width--you may want to experiment by routing PWR and GND both with Real Width turned ON as well as OFF. Enlarge Display until two to three IC's are shown.

Let's review the ROUTE option. When ROUTE is selected, the menu changes to the following:

| | | | | |
|---|---|---|---|---|
| **1 Rut Conn** | **2 Auto Rut** | **3** | **4 Shove** | **5 Modify** |
| **6 Sel Tear** | **7 Unroute** | **8 Len Of** | **9** | **10 Exit** |

1 RUT CONN is used to interactively route a connection (without Design Rules Check).

2 AUTO RUT contains the three Autorouters--Memory, Power/ Ground and Maze.

3 SHOVE contains the Push and Shove Autorouter.

4 MODIFY contains the features used to Interactively modify a track.

5 SEL TEAR is a toggle used to turn Tear Drop pads On and Off of a track.

6 UNROUTE converts a track back to a connection.

7 LEN OF measures the Length of a track or segment.

To interactively route Connection Number One, be in ROUTE (F5), place the cursor over Pin 20 of U17 and select connection Number One using RUT CONN (F1). As you pull cursor away from the pin, the track will follow behind the cursor, at 50 mil Width. Insure that you are on Layer 2. Use the routine of LEVEL (F4). Also, note the Routing information displayed at the left of the screen display in the pop-up window. Make a corner and proceed until you are on the terminating pin (U17.20). Use FIN NET (F7) and the track will terminate on the terminating pin.

Using UNROUTE (F7), unroute Connection One and reroute until you get the hang of Corner, Spacing, etc. Your track should be as displayed as for Connection Number One on Layer 2 of Figure 4.2.

How far should your Vertical Power/Ground track be from the IC pins and from the other PWR/GND tracks on the opposite row of IC pins? To calculate this, you need to develop a concept called "doing your Numbers." This means calculating the number of routing grids or channels between the track center line and the adjacent pad center or track center. Let's consider the spacing between the IC pad and the PWR/GND track. The formula is 1/2 pad diameter + spacing + 1/2 track width. This is 1/2 x 50 + 13 + 1/2 x 50 = 68 mils. You are using a 25 mil routing grid, the next higher increment of a 25 mil grid is 75 mils, therefore, your track center line must be a minimum of 75 mils or 3 grids away from the center of the IC pad.

How far must your track be from another adjacent Power/Ground track? The formula 1/2 track + space + 1/2 track = 1/2 x 50 + 13 + 1/2 x 50 = 63 mils. Again, a Power/Ground track (at 50 mils) must be 3 routing grids or 75 mils, away from an adjacent Power/Ground track.

From the above, we see that as long as your Power/Ground track is 3 routing grids away from the IC pad center and 3 grids away from an

adjacent parallel Power/Ground track, you will meet 13 mil space clearance requirements.

There is another technique you may use in Interactively Routing. Let's try it on Connection #1. Unroute Connection #1 and display it as a connection. Use ROUTE and RUT CONN (F1), however, do NOT have the cursor over the beginning pin at the start of the routing sequence. Instead, have it over the connection itself. The track will start from the Beginning pin as before and follow the cursor, and the connection will extend from the cursor to the End pin. In addition, there are three ways to finish or complete the track: FIN NET (F7), END (F8) and FIN PIN (F9).

Let's try them all. Start routing Connection #1. Note the track from C18.2 to the cursor and the connection going on to pin U17.20.

Try FIN NET (F7). Notice the FIN NET menu is inactive.

Try END (F8). Notice the track is terminated with a Standard Thru Via and a connection exists between the Via and the End pin. This connection can be subsequently routed with RUT CONN or the Auto Routers.

Try FIN PIN (F9). Notice that when used, the track proceeds to the nearest point to the End pin, makes a 90° turn and finish on the End pin.

**Caution**: Always insure there are no intervening tracks or pads between the cursor and the End pin when FIN PIN (F9) is used.

Route Power and Ground connections 1, 2, 3 and 4. UNROUTE and REROUTE as well as MODIFY until you are at ease with the ROUTING routine, especially when routing wide tracks. At the end, your Power and Ground tracks should appear approximately as shown in Figure 4.2, Layer 2.

Let's now route the three Signal connections (signal connection 5, 6 and 7). These connections are to be tracks with a width of 12 mils. Let's do the "Numbers" and calculate their minimum center line distance from the IC pad center. The formula is 1/2 the IC pad (30 mils) + spacing (13 mils) + 1/2 the signal track width (6 mils). The formula, therefore, is 30 + 13 + 6 = 49 (use 50). We must, therefore, route the signal

tracks a minimum of two system grids (2 x 25) away from the IC pad center, in order to clear IC pads. The formula for the signal track-to-track separation is 1/2 the track width (6) + spacing (13) + 1/2 track width (6) = 25 mils (6 + 13 + 6 = 25). We can, therefore, route Signal tracks 1 grid, or 25 mils from other Signal tracks.

Let's start routing Connection Number Five. Connection Five represents a memory connection between two IC's. Start routing the connection at U18 pin 14. Route the track horizontally on Layer One until the track interferes with pin 7 of U18. Add a corner and proceed around the pad, going either above or below the pad, two grids so as to clear the pad. For the corner, use two each 45° corners, since, generally, memory-like tracks, such as Connection Five, use 45° corners. Once clear of Pad 10 of U18, add another set of 45° corners to bring the track back to its original routing channel. Use FIN PIN (F9) (or FIN NET) to finish the track at pin 14 of U19. The completed track should be as displayed in Figure 4.2.

Suppose you wish to partially route a connection. How do you do this?

Convert Connection Number Five back to a connection, using UNROUTE (F7). Now start routing Connection Number Five again. After routing for a distance, finish the track with END (F8). Note the following:

1. The routed portion of the tracks remains as routed.

2. A Thru Via is placed at the end of the track (called Partial Track Via).

3. The unrouted connection from the Partial Track Via to the end pin is presented. This connection can be routed later.

Connection Six is used to illustrate the use of Vias or Feedthrus. Start routing Connection Six by starting the track at pin 12 of U18 on Layer One. Go horizontally until midpoint between the two rows of IC's, add a 90° corner and start the track downward in the -Y direction. Put in a Via at the corner by selecting LEVEL. The track is now on Layer 2 and should be the color selected for Layer 2. Continue the track downward until it crosses below track 5 and is opposite its own terminating pin. Add a corner and move the track to the right toward the terminating pin. Add a Via at the last corner so the track is now on Layer 1 and complete

the track on Layer 1 at pin 3 of U18. The track should be approximately as shown in Figure 4.2.

Now for the last connection, i.e., Connection Seven. This connection illustrates the use of Vias as well as placing a track as close as possible to an adjacent track. How close can a signal track be to a via? The numbers show a track-to-via spacing of 50 mils (27.5 + 13 + 6 = 46, therefore, use 50) and clear Vias using two routing grids. While routing Connection Seven, locate it on the Vertical, Layer 2, as close to previous routed track Six as the spacing rules permit.

The routing of Signals 5, 6 and 7 should be approximately as shown in Figure 4.2.

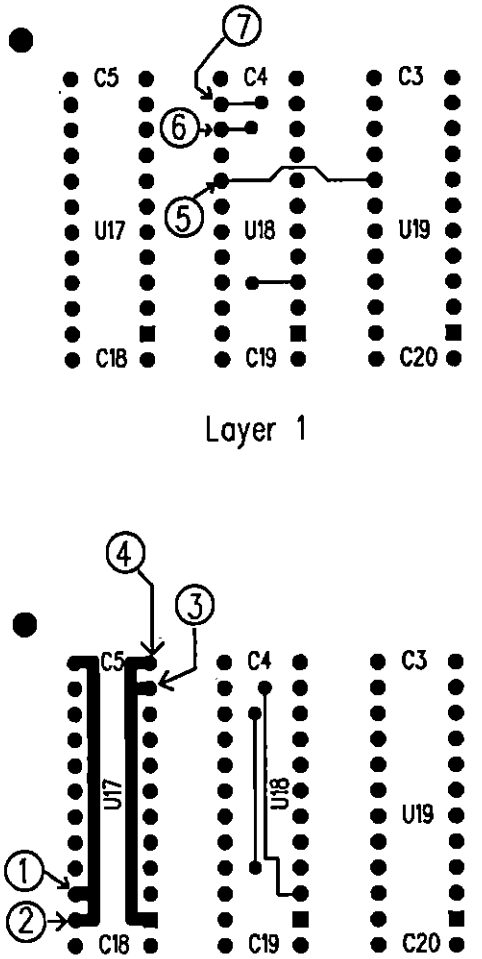



**Figure 4.2**

*Figure 4.2 - ROUTED FILE (TROUTE.JOB)*

You now have the basics of Interactive routing. The next step is to build up **speed**. Speed of routing is very important since approximately 50%-60% of the total time spent designing a board is in the routing phase and, therefore, you need to develop your skills in this area until the use of the Routing subroutines is second nature and very fast. Here's how to do this:

Convert the seven tracks you have just routed back to Unrouted connections using UNROUTE. At this point, you will know the Routing subroutines and the path the tracks will take. Let's now work on speed!! Route and reroute the seven connections until you can consistently route all seven, in 15 seconds or less. When you can do this, the use of the Routing subroutines will start to become second nature, in that you don't have to think about which subroutine to use for a particular function and are well on your way to building up speed in the Interactive routing of a board.

## 4.2 MODIFY TRACK

It is as important to have good skills in modifying tracks as it is in the original routing of the tracks. Here's how you can build up speed in modifying tracks:

Convert the seven tracks back to connections; then route each as a point-to-point track directly from one pin "as the crow flies" to the terminating pin. This is best done by selecting the connection and then COMPLETE on Vertical or Horizontal connections. On Diagonal connections, select the connection, then use Angle, pull the track to its Terminating pin and use FIN PIN (F9). This will result in a point-to-point TRACK or ROUTE which you can Modify. Now, using the various Modify subroutines, including Add Corner, Move Corner, Cut Segment, etc., modify the point-to-point tracks until they are routed approximately as shown in Figure 4.2. You should work at Modify Track, building up your speed until you can consistently modify the seven tracks as discussed above in 45-60 seconds or less.

HINT: You may find that using REAL WIDTH may or may not speed up system performance during MODIFY TRACK. Try both approaches. Also note that MOVE CORNER has both an Orthogonal and Angle feature. Both are very powerful and you need to recognize when to use each one.

## 4.3 T-ROUTING

T-Routing is a form of Interactive Routing that is more efficient, in most instances, as compared to conventional Interactive routing. This is due to the fact that T-Routing permits the starting of a track (not a connection) on another track or ending on another track.

The various functions that can be performed using T-Routing are:

1. Pad-to-Track (within the same Net)
2. Track-to-Pad (within the same Net)
3. Track-to-Track (within the same Net)
4. Pad-to-Track (Not in the same Net) (ECO or On-the-Fly)
5. Track-to-Pad (Not in the same Net) (ECO or On-the-Fly)
6. Track-to-Track (Not in the same Net) (ECO or On-the-Fly)

NOTE: (4), (5) and (6) require changes to the connectivity database and, therefore, can be performed only while in ECO (an ECO file will be created) or ON-THE-FLY (an ECO file will NOT be created).

Typically, how T-Routing is used is the Designer will Interactively route the primary or main connection within a multi-connection Net. Following this, the individual connections within the Net will be tied directly to the primary track at the nearest point on the track using the T-function, thus saving the time to route to the connection pin and also saving board space.

Select the file TROUTE.JOB and display. Window down to the bottom of the board and observe the three connection net running from P1.17 to U9.13 to U8.13 to U2.6 with the U9.13 to U8.13 track already routed.

Using ROUTE, select the connection starting from U2.6 and route toward the nearest point on the track between U9.13 and U8.13 (not to the pin). When routed to the track, use the T-Route feature to tie into the track. This is done by selecting FIN NET (F7) (Finish the Net). You may notice (1) a feedthru is added to tie the two connections (tracks) together (Not necessary if T-Junction is on same layer), (2) the connection being routed is terminated at the track, and (3) the remaining part of the connection is erased--all resulting in a classic T-Routing junction.

NOTE: When T-Routing, always pick the connection near the start pad, NOT the pad itself. This will bring up the menu item FIN NET!

Next, start routing the connection starting at P1.17 and tie it into the nearest routed track, using FIN NET (F7).

Got the idea? Let's try again.

Delete all three tracks back to connections. This time route the U2.6 connection first, then "T" into it from U8.13 using the T-Routing feature FIN NET (F7), then "T" into that track from P1.17.

Let's T-Route in reverse, i.e., go from a track to a pin. Convert the three tracks back to connections, then route the track between U8.13 and U9.13. If the connection doesn't go from U8.13 to U9.13, don't worry. Just route if that way anyway, and it will take that form--that is one of the major advantages of T-Routing, i.e., the ability to reorder the Nodes within a Net. Next, put the cursor over the routed track (between U8.13 and U9.13) at any point and select RUT CONN (F1). You will notice a track started on the track. The track you are routing will be completed on U2, pin 6 using FIN NET (F7) and the connection between U2.6 and U9.13 will disappear. Next, start another track from the first track and FIN NET when over P1.17.

Let's try routing a track from an unconnected pin to the track and then T-Routing into the track. Start at U4.6 and select FIN NET (F7).

Surprised! The track did not start and you received a prompt message as follows:

**Can't route from/to unconnected pin**

The reason for this refusal of your command and the "prompt message" is that you were changing the connectivity database, however, you were not capturing the changes as an Engineering Change Order. The software is "duty bound" not to change either the connectivity or Parts List unless captured in an ECO File. If you deliberately want to change the connectivity, without an ECO, use "ON-THE-FLY."

So what can you do in the above situation:

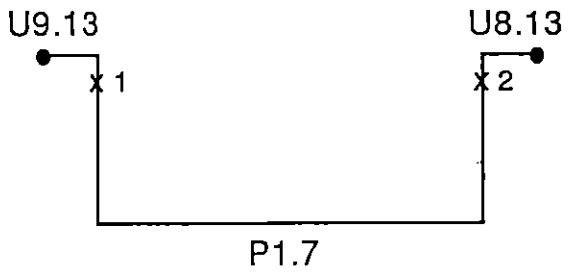
1. You can override the software and take the responsibility for incorporating a change that will be undocumented. To do do this, go to IN/OUT, ONTHEFLY, and ADD RUT. Start on U4.6 and connect to the track with COMPLETE (F9). Respond to the request for new net name.

2. To create an ECO file while adding its route, be in ECO, TO SCH and give an ECO File name. Example: Be in ROUTE, place the cursor over pin 12, U1 and select ADD RUT (F9) to start the track. Route the track until directly on the routed track and select COMPLETE (F9). The track will be completed and an ECO file will be created that adds U1.12 to the Net. (You will be asked for the combined Net Name.)

T-Routing may be used to tie two different Nets together without using the pins within the nets. Create an additional net on the board by routing a track between U12, pin 1 and U16 pin 1. Next, route the P1.17, U8.13, U9.13 Net. This results in two separate nets on the board. Now tie them together with two T-junctions. Using ADD RUT, start at the first net, route to the second net and use COMPLETE. Respond to the prompt:

**Name for the combined net**

and the two nets will be tied together. T-Routing can be used to reduce or clean up an already routed track. Convert the Net P1.17, U9.13, U8.13 back to a connection. Next, route the U8.13 connection in a big "U" down across the board, thus:




Here is a convenient way to clean up this track. Use RUT CONN and "T" a track between points 1 and 2. Note how the "U" is cleaned up automatically.

Finally, T-Routing can eliminate the inconvenience of a "connection" oriented database where one must route a track exactly from one pin to the next pin in the Net, even though a different pin in the Net may be more desirable. T-Routing permits one to route from one pin to any other pin anywhere in the Net. This is especially useful when routing Power and Ground Nets.

## 4.4 THE SHOVE ROUTER

Prior to routing the Training Board, you should become familiar with the SHOVE Router. This Autorouter's features act to "push and shove" existing tracks aside in order to route a connection. It has three modes of operation:

(1) You select the connection and the Router autoroutes it and presents the track at the CRT for your concurrence (AUTO PATH F1).

(2) You select the connection and each corner of the track and the Autorouter autoroutes between each corner (AUTO STEER F2).

(3) You route the connection interactively, however, Design Rules Check prevents making errors (MAN PATH F3).

Let's get acquainted with SHOVE. Have the TROUTE file displayed in an unrouted condition. (Use Figure 4.1 for reference.) Select ROUTE, then SHOVE (F4). A short 5-10 second "Working" message will occur that is used for mapping the area to be routed, after which you will see a White rectangle (the Routing Window) within which Auto Shove Routing will occur. You are now ready to route.

Place the cursor over the connection at Pin 20 U17 and select AUTO PATH (F1). In a second or so (longer on very difficult tracks), you will be presented with the proposed track pattern. If you agree, select CONFIRM (F1) and the proposed track is finalized. Try it with connection #2. Continue until all seven connec- ions are routed. Notice the Pushing and Shoving action while autorouting.

Now unroute all the tracks using UNROUTE (F8).

What if you did not agree with the proposed track pattern or suppose your circuit required a track pattern different from the proposed pattern? You use either AUTO STEER or MANUAL.

Be in SHOVE. Place the cursor over the connection at Pin 20, U17 and select AUTO STEER. Use LEVEL (F4) to go to Layer 2. Pull the cursor to the first corner and select CRN/FIN (F1). Now select LOCK CORNER (F3) to lock the corner down.

After each corner, lock the corner down. Use COMPLETE (F9) to finish the track. Make the routed track "Shoveable" when routing succeeding tracks by selecting SHOVE (F1) or Unshovable by selecting NO SHOVE (F2).

The concept of AUTO STEER (F2) is to let you "Steer" the pattern while the Autorouter pushes and shoves other previously routed tracks to make room. When using AUTO STEER, you must:

(1) Lock down a corner immediately after putting in the corner (otherwise the corner will move).

(2) Finish the track with COMPLETE (F9).

(3) Do NOT "force" the Autorouter to take a path significantly different from its normal Auto Route Path which follows "as the crow flies" between the two end pins. In other words, don't expect the Auto Steer Router to assist you in taking a circuitous route between the end pins.

Now AUTO STEER the remaining connections on the board.

NOTE: Normally you would not use AUTO STEER to route all the connections. Instead, use AUTO STEER only for those critical nets where you wish to control the Autorouter pattern.

We will now route these seven connections using SHOVE Interactive Routing with On-line Design Rules Checking . Convert all tracks back to connections. Be in SHOVE, place the cursor over the connection at Pin 20 U17 and select MAN PATH (F3). The connection will be selected. Use LEVEL (F4) and insure you are on Layer 2. Pull the track to the first cursor location and put in a corner with CRN/FIN (F1). Pull the cursor to the next corner and put in a corner, etc. When ready to finish, use COMPLETE (F9). Use SHOVE (F1) to permit shoving

of the completed track or NO SHOVE (F2) if you do not want it to be shoved.

MANUAL PATH of SHOVE will NOT permit you to make a spacing error. Let's try it. Start routing the next connection with MAN PATH and try to create an error by routing the track over a pad or track. Notice that you cannot create an error!

Now complete the other connections using MAN PATH.

## 4.5 ROUTING THE TRAINING BOARD

When you feel comfortable with your ability and understanding of the Routing Routine, you are to route the training board.

Call up file XXX7, the file representing the final placement as created in the previous chapter. You will route its Power and Ground connections first, followed by the Memory array signal connections, and finally the remaining Signal connections. In the process we shall maximize the use of the Autorouters in PADS-2000. Have the HIGHLIGHT feature of PARAMETERS ON.

### 4.5.1 PLACEMENT/ROUTING GRID SYNC TEST

Before using any Automatic Router, or the Interactive Router, check that the Routing Grid and the Placement Grids are coincident or in sync. If during the course of placing the board you have had reason to set the Grid to a low value such as 1-5, and also reset the Origin, the placement and routing grid may have gotten out of coincidence. To check, have the grid set to the Routing Grid and using SETUP, ORIGIN, reset the Origin. Observe the prompt message for the new origin location. Are the X and Y location XXX?? and XXX?? values such that they are multiples of the Routing Grid (Example: 00, 25, 50 75 if using a Grid of 25). If so, then the two grids are in synch. Proceed to route. If not, for example (?? = 02, 12, 09, etc. when using a Grid of 25), then the two grids are not in Sync. DO NOT TRY TO ROUTE. INSTEAD, use the procedure outlined in Section 4.5.6 to correct the situation.

## 4.5.2 POWER AND GROUND ROUTING

Using SETUP, NET ATTRIBUTE, make the Power (+5V) and Ground (GND) connections only, visible. Other connections should be turned OFF so that the connections displayed at the screen are the Power and Ground connections only.

Under SETUP, PARAMS, set Net Highlight ON, Real Width to 1, Pad/Pad 13, Pad/Track 13, Track/Track 13.

In NET ATTRIBUTES, Set Power and Ground to Display ON, LENTYP to Vertical and ALL to Off. Only Power and Ground connections should be visible at the Display. Run a Length Minimization (F6), PLACE, AUTO, LEN MIN (F7). They should be oriented vertically on the board for ease in routing under the IC's on Layer 2.

If there are connections on diagonals across the Decoupling Caps, thus:

there is a problem that will need fixing. This condition occurs because the pad 1 and pad 2 (PWR and GND) of the Decoupling Caps are not in line with the Power and Ground pins of the IC's. You will need to rotate the Decoupling Caps 180°. It's simple to do.

Unglue all caps: **UNGLUE C* <CR>**

GLUE C1 and C2

Rotate ALL Decoupling Caps: **ROTATE C* <CR>**

Again, Rotate ALL Decoupling Caps: **ROTATE C* <CR>**

We will need to move the Decoupling Caps back into position. First in PLACE AUTO, MATRIX, UNSELECT the matrix used for Auto Rename and SELECT the matrix used originally to Auto Place the Decoupling Caps. Using MOVE, move the matrix until the matrix points are EXACTLY above the Right row of IC pins. The matrix point will be the location of Pin 1 of the Decoupling Caps after Auto Place and

will result in the Power pin of the Decoupling Pin being directly above the Power Pin of the IC's.

Before running INITIAL PLACE, be sure all parts except Decoupling Caps are Glued, otherwise you will ruin your entire placement.

Glue All: **GLUE *<CR>**

Unglue Caps **UNGLUE C* <CR>**

Glue: **C1 and C2**

Test to insure these conditions.

Just to be on the safe side, make a Job Save XXX8.JOB.

Select PLACE, AUTO and INITIAL PLACE (F4). The Decoupling Caps will be relocated on the matrix.

Double check to see if any Decoupling Cap pads overlap IC pads. If so, use PLACE, MOVE to relocate the Decoupling Caps.

Always, but always, run a Space Check with Errors Visible in Display to insure there are no Pad/Pad errors after Placement and before Auto-routing.

Now rerun Length Minimization (PLACE, AUTO, LEN MIN). The diagonal connection at the Decoupling Caps should now be removed and the Power and Ground Router can route the most of those connections.

Now we will route all Power and Ground connections on the board, taking care to keep Vertical tracks inside the IC pin rows on Layer 2 and Horizontal tracks on Layer 1.

When routed, the Power and Ground tracks should be approximately as shown in Figure 4.3.

With only Power and Ground displayed at the CRT, we will use the Autorouter to route Power and Ground.

In case you do not like the first attempt at Routing, make a save XXX9, that you can return to.

Check all your parameters including Grid, and select ROUTE, then AUTO RUT (F2).

- Respond 25 to Routing Grid
- Respond 50 to Via Grid
- Select SETUP (F7) and select only Short P/G and Long P/G in the pop-up window. All other entries should be deactivated.
- Select Board (to route entire board)

The Power and Ground connections will be autorouted with the P/G Heuristic Router. The window (upper left) indicates the status. The prompt line will display the number of connections routed and the number unrouted.

You need to appreciate the following features about the Power and Ground Heuristic Router. The router is extremely fast because it does not search to find a track position. In fact, it considers only two considerations:

(1) Is the Connection exactly Vertical or Horizontal--if not, it will not be considered by the Router, and

(2) Are there any obstructions that prevent the connection from being made with a three-segment track, thus:

If there are any obstructed connections, the connection will not be routed.

(3) Any connections that are turned ON in NET ATTRIBUTE will be routed, regardless of whether or not they are actually PWR and GND, therefore, turn PWR and GND ON and ALL to OFF in Net Attribute, before using the Power and Ground Router.

(4) The router will route on one layer only, with no Vias.

(5) Copper

a. The Router will NOT route a connection that crosses a copper line.
b. The Router will NOT route a track that crosses the edge of a copper polygon.
c. The Router WILL route a track whose end points are totally contained within a copper polygon.

(6) Short Power/Ground routes connections equal to or less than 0.6" in length. Long Power/Ground routes connections over 0.6" in length.

Upon completion of Power and Ground Autorouting, all but a few connections should have been routed. These were not routed because (1) they were not exactly Vertical or Horizontal, or (2) they were blocked from routing.

Let's use the SHOVE Router and AUTO PATH to route the remaining connections.

Select ROUTE (F5), then SHOVE (F4). Place the cursor over one of the unrouted Power and Ground Connections and select AUTO PATH (F1). The connection will be routed. If you concur, select CONFIRM (F1).

NOTE: Connections within the Routing box only will be routed. Use the keypad functions to relocate the routing box.

Repeat for the remaining Power and Ground connections.

In the event there are cross ties between rows of IC's, they may not route if the pads are so close that a 50 mil track cannot route between the pins. If so, route at the top of the rows of IC's, as follows:

(1) ROUTE, RUT CONN and ADD put a track between the Power (or GND) pins at the top of the rows of IC's.

(2) Run LENGTH MINIMIZATION. (Be sure LEN MIN in NET ATTRIB is set to Vertical and the cross tie connections will disappear.)

Carefully inspect the entire board to insure that there are no P/G connections remaining unrouted. Save the routed file as XXX9.JOB.

## 4.5.3 SIGNAL HEURISTIC ROUTER

We will now route the connections that lend themselves to routing by using the Signal Heuristic Router of Autoroute. These connections consist of the Signal connections that are Horizontal or Vertical or "Near" Horizontal or Vertical.

Using NET ATTRIBUTE, set ALL connections to Display ON and LENTYP to MIN. Set +5V and GND to Display OFF. Exit and only the Signal connections (not PWR and GND) should be displayed.

Select ROUTE, then AUTO RUT (F2).
Grid = 25
Via Grid = 50

Using SETUP (F7), turn Power and Ground to OFF and Short Memory and Long Memory to ON.

Select BOARD (F4) and the Signal Heuristic Router will route the Memory array.

Save the file containing the routed Power and Ground and Memory connections as file XXX10.JOB

The finished Memory tracks (and Power and Ground tracks) should be approximately as shown in Figure 4.3.

A subset of the Signal Heuristic Router is HORIZ/VERT, SHORT ONLY, and LONG CHANNEL routers. We will now route these signal connections that can be routed with HORIZ/VERT, SHORT ONLY and LONG CHANNEL.

Select ROUTE, then AUTOROUTE.

Grid = 25
Via Grid = 25

Using SETUP (F3), set SHORT MEMORY and LONG MEMORY to OFF. Set HORIZ/VERT, SHORT ONLY and LONG CHANNEL to ON.

Select BOARD and this category of connections will be routed.

Save the file as XXX11.JOB.

NOTE: The Signal Heuristic Router will not route between pins on a component that are spaced closer than 100 mils.

## 4.5.4 SIGNAL MAZE ROUTER

We will now use the Maze Search Router of AUTOROUTE to route the remaining signal connections.

Select ROUTE (F5), then AUTO RUT (F2)
Routing Grid = 25
Via Grid = 25

In SETUP (F7), turn Short Memory and Long Memory to OFF.

Select "one Via" thru Six Vias. The Maze Search routing passes will be run.

The results of each routing pass will be displayed in the upper left window and in the prompt line.

Upon completion of Routing, there probably will be in the order of 10 to 20 connections remaining on the board that were not routed, the majority of these connections appearing at the edge connector.

Use the SHOVE Router to shove aside those previously routed tracks to make room for the unrouted connections.

NOTE: When routing Signal connections in the Shove Router, Always have ALL tracks turned ON. The Autorouter must "see" the routed tracks, otherwise it will assume the space is unoccupied and will route into the space. In SETUP, NET ATTRIBUTES, set ALL, +5V and GND to ON. Check the PWR, GND and Signal tracks are displayed.

NOTE: If PWR and GND are Not turned On, then the Shove Router does not "see" the undisplayed PWR/GND tracks, thus there may be space errors.

Select ROUTE (F5), then SHOVE (F4). Place the cursor over any unrouted connections and select AUTO PATH (F1). When the connection is routed, select CONFIRM (F1).

Continue to use SHOVE until all the connections have been routed.

NOTE: In those rare instances where SHOVE will not complete a connection with the first attempt of AUTO PATH, immediately without moving the cursor, make another attempt. If not successful on the second and third attempt, you must use your skills as a manual designer, together with Manual Route to complete the connection. It is highly probable that there simply is not enough room to manually route the connections. In fact, after using SHOVE on all connections, and if there are still a half dozen or more connections still unrouted, you need to replace the board or get relief on the board size.

NOTE: Difficult routes may take 1-2 minutes or more, but will always be faster than you can modify tracks, and route the connections.

To check if all connections are routed, use REPORTS and STATS (F4) and make a printed report. Examine for Unrouted or Partially Routed connections.

To Locate: Turn tracks to OFF and inspect.

Partially Routed connections will result when using SHOVE, AUTO PATH and the connection extends outside the routing window. The connection within the Routing Window will be routed within the Routing Window but NOT routed outside the Routing Window.

### 4.5.5 VIA MINIMIZATION/COPPER POUR

When all connections are routed, use AUTO ROUTE and the VIA REDUCE pass to remove vias from the finished routed board.

The Design Criteria requires a poured copper plane on Layer 1 under Y1, R1 and R2 and their associated tracks, using 15 mil space clearance.

In PARAMETERS and SELECT, set COPPER POUR CLEARANCE to 15. Make sure you are on Layer 1. Enlarge the area containing Y1,

R1 and R2 in CREATE, COP POUR (F6). Select NEW POLY (F1) and draw a polygon enclosing Y1, R1 and R2 and their tracks. Select FLOOD (F8) and observe the polygon being flooded with copper.

The poured copper polygon is to be connected to GROUND. Using COP POUR or COPPER, MODIFY, and NET NAME (F5), give the Poured Copper area the name of GND. Next, turn On GND in NET ATTRIBUTE and using ROUTE and MODIFY (F5), modify a GND Track such that it runs thru or touches the Copper area on Layer 1.

NOTE: After routing a track through Poured Copper, always reflood the copper in order to insure space clearance around the track if not at the same potential.

Make a Job Save as XXX12.

### 4.5.6 ROUTING GRID SHIFT

The Routing Grid as well as the Placement Grid is established by the System Grid Setting and the Origin location. Moreover, the two Grids must always originate from the same origin point, otherwise the Autorouter will not route as expected, due to the Routing Grid Shift relative to the Placement Grid.

Normally, when placing the parts on a board, a Placement grid is used that is coordinated with the Routing grid. For example, if you expect to route on a grid of 25, you would place the parts using a grid of 25, 50 or 100 rather than a grid of 10, 12.5, 16.67, 20, etc. In such an instance, if you do not change the grid and relocate the origin, there will not be a problem with a shift between the Routing grid and the Placement grid. In fact, even if you relocate the origin but do not change the grid, a problem will not be created.

The problem is created when, after placing the parts with a grid of, say 25, and a small grid, such as G = 1, 2, 5 is selected and, while selected, the origin is relocated, following which, the grid is set back to the original placement routing grid.

What has happened is in the above process the placement grid and the routing grid have been forced out of "synch," thus when a track is

routed, it will be off the intended grid by whatever the "out of synch" amount happens to be.

It is important for you to recognize such a situation should you ever inadvertently cause the situation.

Here is how you check for such a situation: Call up your Training board placed, but before Routing has occurred. Make sure the Grid is set to your placement grid (100, 50 or 25).

Using SETUP, place the origin in 3-4 different locations. Notice that the prompt:

**New origin is at System Location XXX?? YYY??**

is always on the system grid. That is to say, the ?? in both X and Y correspond to multiples of the placement grid (if the Placement Grid is 25, then ?? will always be 25, 50, 75 or 100). This means your Placement Grid and Routing Grid are in synch and the tracks will always be on a 25 mil grid. In fact, to visually see this, Interactively route a track Horizontally between 2 IC pins and zoom up the screen. Notice the track is exactly in the center between the pins, indicating a 25 mil Routing Grid in synch with a 25 mil Placement Grid.

Let's now force the Placement Grid and the Routing Grid "out of sync" and observe the results. Set the Grid to a Grid of 1. Relocate the Origin. Notice the prompt:

**New origin is at System Location XXX?? YYY??**

no longer has the ?? as multiples of 25. Instead, ?? can be any number between 00 and 25. Example: 36215 16510. The 36215 indicates the routing grid has been shifted 15 mils above the 36200 grid while the Y grid 16110 has been shifted 10 mils above the 16100 grid.

What happens if you were to try to route a track between the 2 IC pins? Remember the parts are placed on a 25 mil grid. Go ahead and try it. Set Grid to 1 and relocate the origin. Notice the ?? is at some value other than a multiple of 25. Try routing the track between pins. Zoom up. Notice the track is no longer exactly between the pins but is offset some distance depending upon the value of ??. If you attempt to

autoroute, in all probability, the Autorouter will not route if the "out of sync" is too large.

If you find yourself in such a situation, what do you do to eliminate the "out of sync" condition? The following is an easy procedure that you should use:

1. Set Grid to 1.

2. Place Origin. (Do NOT move the mouse.)

3. What are the values of ??. Let's say they are XXX15 YYY10. You will then want to shift the grid -15 in X and -10 in Y.

4. Set Grid to 25 (or whatever the Routing Grid happens to be).

5. Use the Modeless command of S -15 -10 <CR>

6. Without moving the mouse, reset the Origin. The ?? should now be a multiple of 25 (assuming a Grid of 25).

7. If not successful, repeat until the Out of Sync value (??) indicates the Routing Grid and Placement Grid are again in synch.

This completes the Tutorial on Routing. With the exception of routing of a multi-layer board, you should be comfortable with the Routing routines and ready for the Checking effort in the next Tutorial.

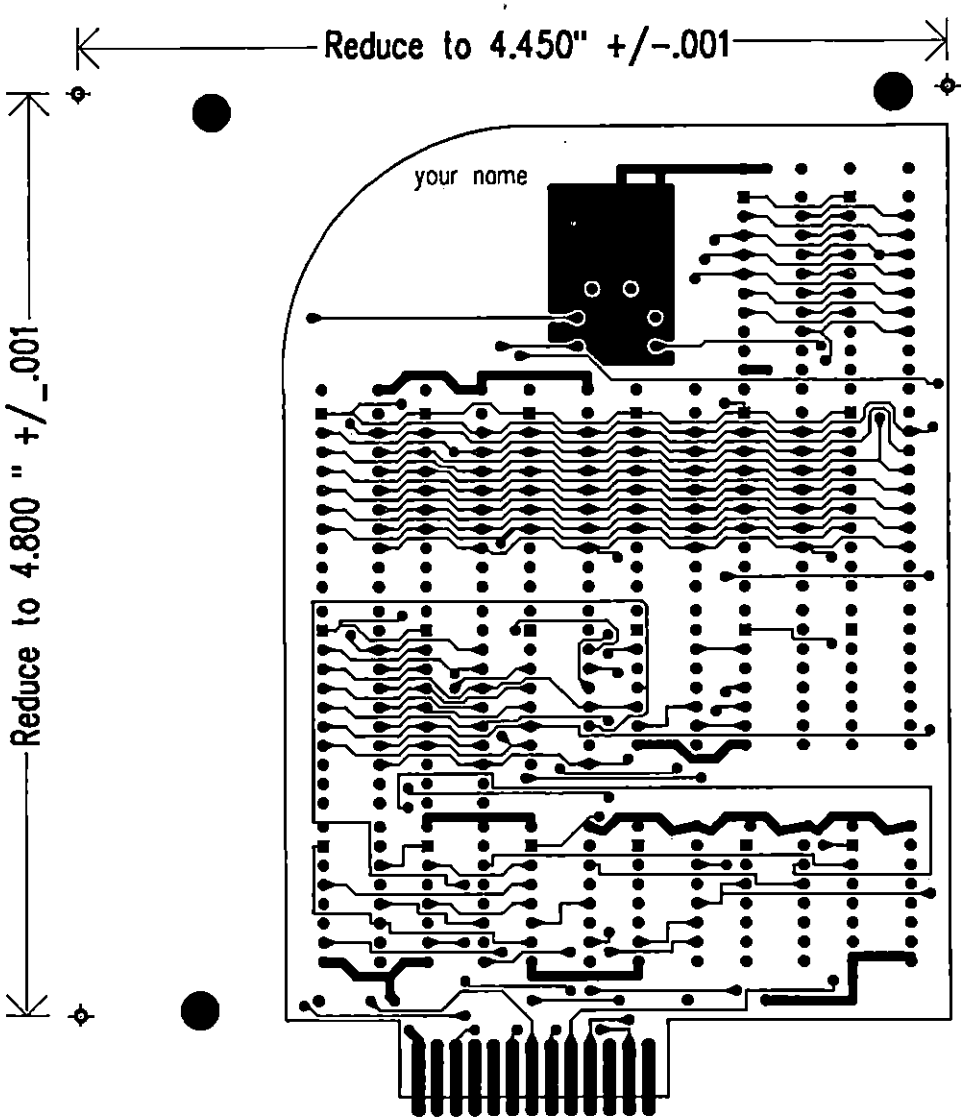



Figure 4.3a

*Figure 4.3a - ROUTED TRACKS, LAYER 1*

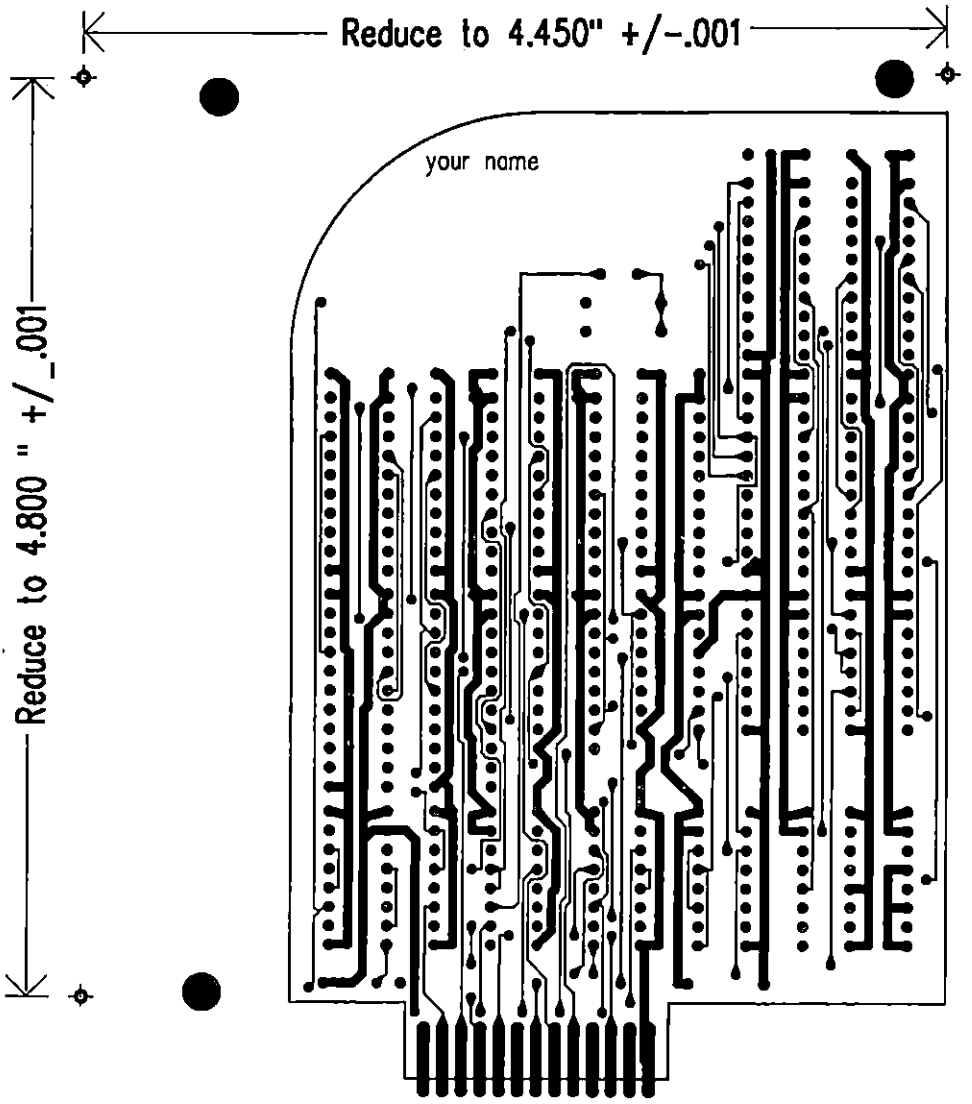



*Figure 4.3b - ROUTED TRACKS, LAYER 2*

## Chapter 5

# TUTORIAL 4 - REPORTS, CHECKING, AND CHANGE ORDERS

(240 minutes)
In this Tutorial, we will consider the use of the Checking Routine, Reports and the incorporation of Engineering Change Orders.

## 5.1 REPORTS

(30 minutes)

The routine REPORTS generates a number of reports that are of value at different times during the design.

At this time, call up your File XXX12 and run the different reports to create a hard copy printout and study them, get familiar with their format and how they would be useful for you at different times during the design.

The LIMITS is of special interest if you are designing a large board, in that it indicates whether the design database is approaching the limits of the available memory.

Review the details of Reports by reading Chapter 10, Volume II, Reference Manual.

## 5.2 CHECKING

(90 minutes)

The CHECK routine examines all of or a part of the board and evaluates the distance or space between pads, vias and tracks, against your preprogrammed spacing criteria. It then reports any violations in the form of error symbols at the screen adjacent to the location of the error.

WARNING: All items (Signal plane, GND, Pads, Vias, Copper) must be ON and Visible at the CRT for the Checking Routine to "see" the items. Take care to have the items displayed before running Checking.

Review the use of Check by reading Chapter 8, Volume II, Reference Manual.

The approach we will use is to run a Space Check on a practice board file wherein we have incorporated some errors for you to analyze and correct. After this check, you will clear the errors. Following this, you will check the board that you routed and saved as file XXX12.

Call TCHECK.JOB up at the screen using the IN/OUT routine and Job In and display at the screen. Using SETUP, PARAMS, set the three check spacings to 13 mils. Set ERRORS, both layers to Visible in the Display menu, and use a color that will contrast with the pads, tracks and copper on each layer. Run a Space Check over the entire board and note the errors. You will notice that there are seven errors that have been put in the board. Study the errors and using MODIFY ROUTE, correct the errors.

NOTE: Space Check checks the area shown at the CRT. To check the entire board, have the entire board displayed. For a quick check of a portion of a board, have that portion displayed.

NOTE: In all probability you will have some errors, resulting from the Tear Drop Pads, since the Router does not consider Tear Drops during routing. You have the choice of:

(1) Turning OFF the individual Tear Drop Pad at the point of error.

(2) Leaving the Tear Drop ON and interactively modifying the track to result in no error.

(3) Ignoring the error; it is approximately 1 mil in error.

CAUTION: In order to display the large number of errors that may result after checking, multiple errors that may occur on a track segment are summarized into a single error. ALWAYS rerun CHECK after clearing errors.

While working on an area of the board, have that portion of the board displayed, run a Space Check to determine if you have cleared all the errors. If not, clear the remaining errors and rerun the check.

When all errors have been cleared from that section of the board, move to another quarter of the board and repeat the sequence. Continue until all sections of the board have been error checked and cleared.

When you believe all the board has been checked, run a check of the entire board, then suppress all items except Error indications. If there are no error indications displayed, you can be confident that no errors of the board violate the spacing criteria.

You should at this time, have a good basic understanding of Space Checking and the clearing of the resulting errors using Route Modify.

Let's now use your knowledge and expertise on the board you have previously routed. Call up your file XXX12 representing the last file you made after routing all the connections on the training board. Set the check spacing size to 13 mils, run the check of the entire board and examine the errors. How did you do? If you had a dozen spacing violations or less, you did very well. Take a section of the board and clear any errors, rerun a check of that area and insure that the errors have been corrected. Continue over the entire board until the entire board has been checked and cleared. Then, as before, make all items invisible except error indications, and run a final check. When there are NO errors in the board, Save as your file XXX13.

In addition to SPACE CHECK, there is a second form of checking with which you should be familiar. This is a "Data Check" of parts and connections in the current Board database against the original input data, either ASCII, FUTURENET, or SCHEMA. Our original input data was ASCII files from PADS-LOGIC, therefore, we would select CHK ASCII (F2), and respond to the prompt with the name of the original ASCII Input file of TPART and TCONN. A comparison would be made between these input files and the current board database. Inasmuch as you have run a Gate Swap, a Pin Swap and a Component Rename, the differences between the input data and the board database would be so great as to make such a data check meaningless unless the schematic were also changed, therefore, do NOT run this data check. Instead, in the future, use the data check to insure the board database

and the input data match, just prior to major perturbations to the data such as Gate/Pin Swap and Component Rename.

TIE PLANE is a check used to determine if all the nodes or pins in a net designated as Nets to be tied into planes, as selected in SETUP, NET ATTRIBUTES, have actually been tied into planes.

To use, select CHECK, then TIE PLANE. Respond to the prompt with the name of the file to be used to list the pins that are not previously tied into planes. The format is as follows:

Unconnected to Plane Layer Report

GND
U7.10 U6.8 U6.4 etc.

+5V
P1.1 U2.14 U3.14 etc.

## 5.3 ENGINEERING CHANGE ORDERS

You should, at this point, read Chapter 9 (ECO), Volume II before starting this section on incorporating Change Orders.

(120 minutes)
We know, of course, the best way to handle ECO's is to not have changes in the first place. Unfortunately, ECO's, like taxes, are inevitable and, therefore, we need to learn how to use the system to incorporate ECO's as quickly and as painlessly as possible, and without errors.

The PADS system permits you to make the change in either the Schematic and then update the board database or make the change in the board and then update the schematic. You will find this flexibility quite useful.

From the Schematic

The PADS-LOGIC Schematic capture has the capability of automatically creating an ECO, when the change is made in the schematic. This change is formatted in the correct format such that when read into the Board database using ECO and FROM SCHEMATIC (F2), the board is instantly updated with the change.

Let's try it. Have File XXX6.JOB displayed at the CRT. This is the Practice board placed before Gate/Pin Swapping and Component Rename.

Select ECO, now FROM SCHEMATIC (F2). The Input Schematic file name is TECO.ECO and consists of the deleting of part U12, 6167 and its associated pins and the adding of Y2, XTAL and its two connections. (Use ALT 9 and TECO.ECO to examine the ECO file.)

When the ECO file is read into the board database, observe that U12 is deleted (but its connection nets are not broken and Y2 (XTAL) is added at the board origin.

To the Schematic

The ECO Routine has all the subroutines necessary for Design Change incorporation into the board. We will use this subroutine to incorporate a design change into the training board. Figure 5.1a, b, c and d, the Training Board schematic, has been marked up to show some design changes that you are going to incorporate into the database of your training board.

Some suggestions for ECO incorporation are:

1. Make a detailed listing of the "Adds" and "Deletes" and use this detailed listing rather than directly using the schematic for cutting in the changes into the board database. Have the detailed listing checked for accuracy before cutting the change in.

2. Your ECO listing should be in the following order for maximum efficiency during the incorporation into the database:

- list the deleted connections
- list the deleted components

- list the added components
- list the added connections
- list all others

3. Incorporate the changes into the board database, using the ECO Routine; then using REPORTS, make a listing of the resulting Connections and Parts and check the listing for accuracy of the changes incorporated before spending any time in routing the changes. Route the changes and Air Gap Check the board after, and only after, the new database has been checked to insure the ECO has been cut in accurately.

Let's get on with the ECO.

Call up the TPRACTIC.JOB File.

NOTE: Figure 5.2 is our listing of "deletes" and "adds". Don't use Figure 5.2, however, as your listing; instead, make your own change listing from Figure 5.1 and compare your list to ours.

Select ECO, then TO SCH. Respond with the name of the ECO file that is to be created to send back to the schematic for updating the schematic. (Use TPRACTIC and a file TPRACTIC.ECO will be created.)

Delete Connections
Using ECO TO SCH AND DEL CONN, use the cursor to select the connection or the pin, delete:

| | |
|---|---|
| P1.4 | U2.5 |
| U3.10 | U3.4 |
| P1.23 | U4.1 |

Using, DIS CONN PIN, disconnect U9.1 from its net and disconnect U6.4 from Ground.

## HINTS:

1. You need to make a hard copy of your ECO. Either use your matrix printer and print out as the change is made or collect the changes in a file in PADS/FILES called TPRACTIC.ECO. View the ECO file. Use ALT 9 and the name TPRACTIC.ECO.

2. Use Modeless Search command of S to search from a node. Example: S P1.4 will move the cursor to pin 4 of part P1, then select (F1) SELECT.

3. Use the "Box Window" command to enlarge the window around the area of interest.

4. After a connection has been picked but before deletion, double check its identification in the Yellow window at the left of the CRT before confirming the delete.

5. Note that when disconnecting a pin in the middle of a net, the connectivity of the net is maintained by a connection put in automatically by the system between the nodes adjacent to the disconnected node (pin). You will need to route this connection.

6. **Most important!!** Insure that WIDTH is set to the desired width of the added connections before the additions (are made)!

Delete Part
Using DEL PART, and the cursor, select and delete R1.

Note that the connection going to R1.1 is deleted since it is at the end of the connection tree. The connection going to R1.2, however, is in the middle of the connection tree and when pin R1.2 was disconnected, the connectivity of the connection net was retained, between Y1, R2 and U2.

Add Connection
Using ADD CONN and the cursor, add connection U2.10 U3.10.

**HINT:** Use the Search command to locate each node using (F1) SELECT.

Using ADD CONN and the keyboard, add connection U3.4 U6.4. Which do you prefer--the cursor or the keyboard?

Using ADD CONN and pick device of your choice, add U2.6 Y1.2.

Add Part
Using ADD PART, add part R3 (same part as R2) and locate in the former location of R1.

To duplicate this position, use KEYBOARD (F2) and Part Type of RES1.

Add Connection
Using ADD CONN, add the connection to R3.1 to P1.23 and R3.2 to U2.2.

Exit the ECO Routine. Now run Reports, NET LIST. Examine the printout of connection changes, paying particular attention to the connections that were deleted, changed or added. Make any corrections as necessary.

Following the checking of the Connection list, run reports PARTS, and examine the printout to insure that the adds and deletes have been incorporated correctly.

Print out TPRACTIC.ECO. This file would be used to automatically update the schematic if the schematic were created in PADS-LOGIC.

When you are satisfied that the database correctly represents the intent of the ECO, run a LENGTH MINIMIZATION. Then, using the ROUTE routine, both Route and Modify, route the connections remaining after the design change is cut into the database.

Following the Routing of the ECO incorporated connections, run a Space Check and correct any spacing errors found in the check.

Deletes:

1. P1.4 U2.5
2. U3.8 U3.4
3. U6.4 GND
4. Part R1
5. Disconnect U9.1
6. P1.23 U4.1

Adds:

1. U3.8 U2.10
2. U6.4 U3.4
3. U2.6 Y1.2
4. R3 (same part as R2)
5. R3.1 P1.23
6. R3.2 U2.2

*Figure 5.2 - ADDS AND DELETES*

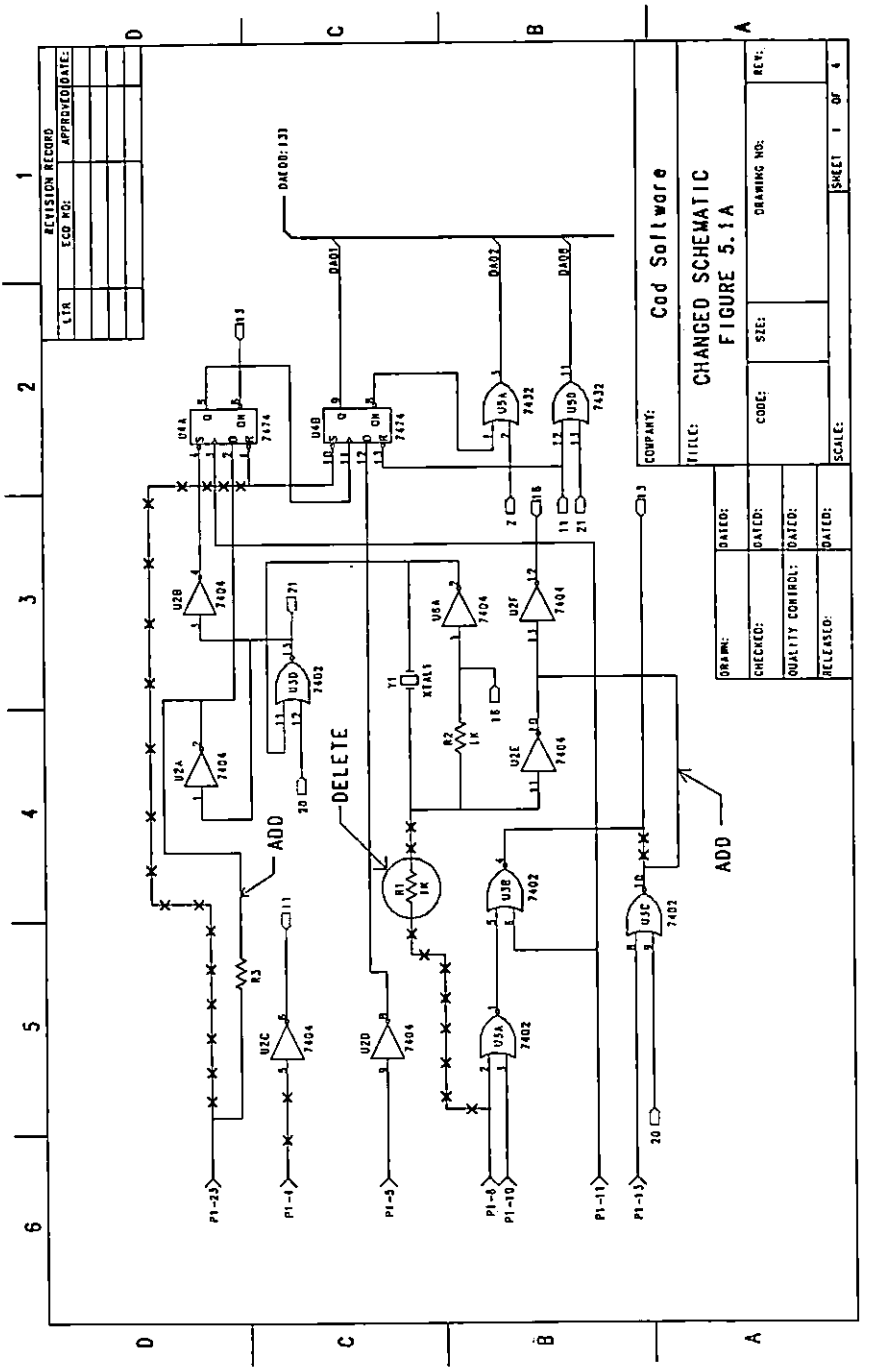



*Figure 5.1a - CHANGED SCHEMATIC, SHEET 1*

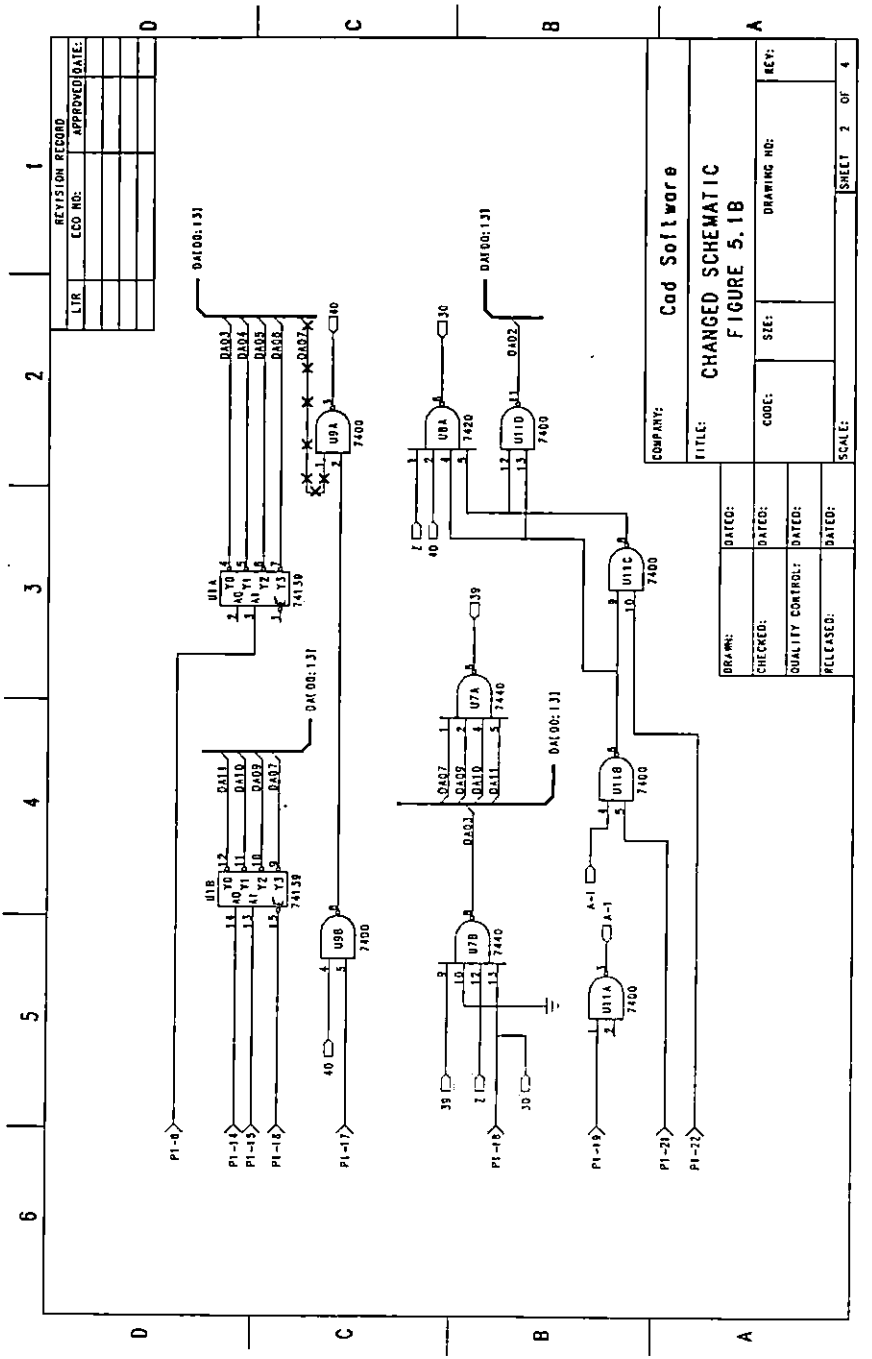
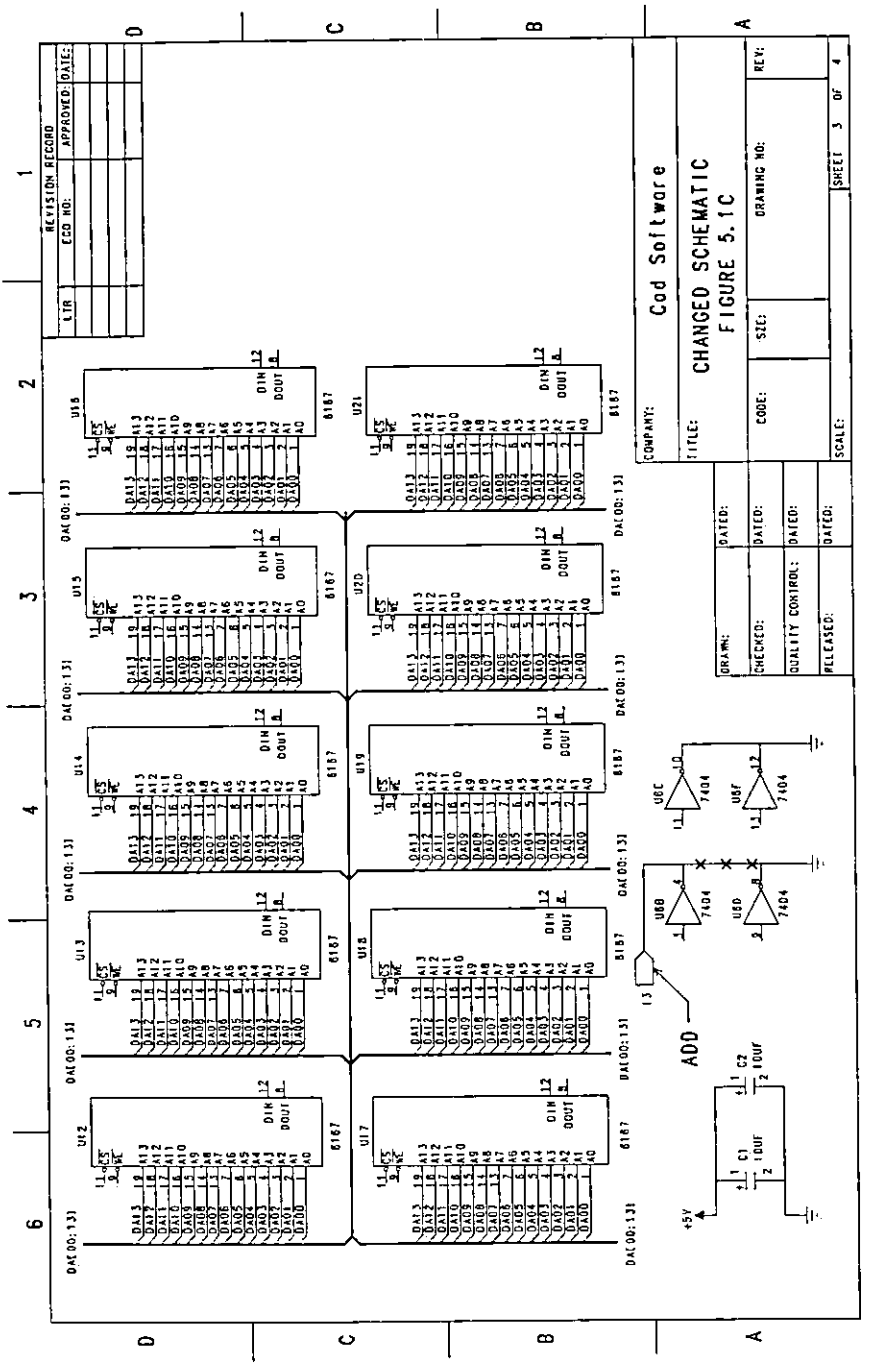



*Figure 5.1b - CHANGED SCHEMATIC, SHEET 2*


REVISION RECORD
LTR
ECO NO:
APPROVED:
DATE:
COMPANY:
Cad Software
TITLE:
CHANGED SCHEMATIC
FIGURE 5.1C
CODE:
SIZE:
DRAWING NO:
REV:
SCALE:
SHEET 3 OF 4
DRAWN:
CHECKED:
QUALITY CONTROL:
RELEASED:
DATED:
ADD
+5V
U12
U13
U14
U15
U16
U17
U18
U19
U20
U21
U22
U23
U24
U25
U26
U27
U28
U29
U30
U31
U32
U33
U34
U35
U36
U37
U38
U39
U40
U41
U42
U43
U44
U45
U46
U47
U48
U49
U50
U51

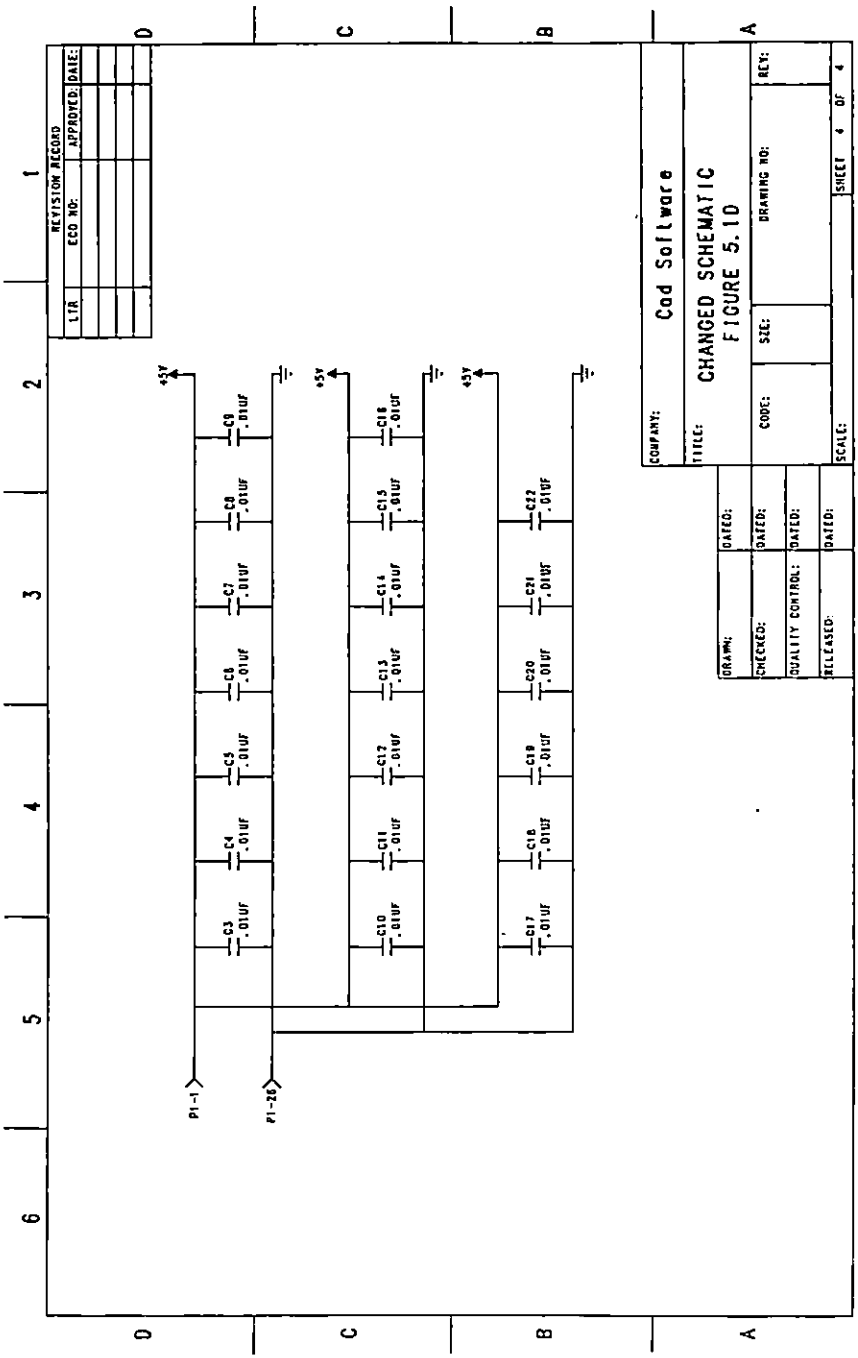



*Figure 5.1d - CHANGED SCHEMATIC, SHEET 4*

## Chapter 6

# TUTORIAL 5 - COMPUTER MANUFACTURING AIDS

*(60 minutes)*
In this Tutorial, using the matrix printer on your system, you will create check prints of the following manufacturing documentation of the board you have designed:

1. Artwork, layer 1
2. Artwork, layer 2
3. Assembly Drawing
4. Solder Mask, layer 2
5. Drill Drawing

Go to Chapter 11 of the Reference Manual, Volume II, and read through the section of the chapter dealing with the procedures for the matrix printer. You will work on the matrix printer with the understanding that the wet ink plotter--Gerber post processing is very similar and can be implemented by yourself, following the exercise on the matrix printer.

Before starting this Tutorial, please make sure that your printer is on-line with paper loaded ready for printing.

You will be using your completed design job file XXX12.JOB for this Tutorial.

Select CAM (F9), to enter the CAM Routine. Upon entering the CAM Routine, the following prompt will appear at the bottom of the screen:

**SPECIFY CAM OUTPUT SUBDIRECTORY**

In response, type XXX12 <CR> (your finished board file name). This will create a subdirectory called XXX12 for storing the files that are created in the CAM Routine that are associated with XXX12. For more details, refer to the Reference Manual.

## 6.1 ARTWORK, LAYER 1

Select DIRECT (F1), which will display the full screen menu.

Selecting the Output Device
Move the mouse cursor over the words "Matrix Printer" and press button #1. Notice how the "Matrix Printer" background changes color. This shows that the Matrix Printer has been selected as the output device. This action of selecting an item from this type of menu will be referred to as "Picking" throughout the rest of this Tutorial. Now "pick" the words "Matrix Printer" again and notice that the box and the words revert back to their original colors. Picking something that is already selected will cause it to become no longer selected. Now pick the words "Pen Plotter" and notice that in addition to selecting the Pen Plotter box, an additional menu has appeared in the blank area on the right side of the screen. For now we will ignore this additional menu, since we are dealing only with the Matrix Printer. So let's move on by picking the words "Matrix Printer" again. Notice how the Pen Plotter automatically de-selects and the Matrix Printer is now selected again.

Now pick the word "PROCEED" and a new menu will be displayed.

Selecting the Plot Type
This menu shows all the different types of plots that can be generated. The first plot we are doing is Artwork, Layer 1, so pick the words "Artwork Plot". The Artwork Plot box is now selected and what was the blank area of the menu now shows all the different plot levels or layers that can be selected. Pick the box with the number "1" for Layer 1 and then pick "PROCEED," which will cause a new menu to be displayed.

Selecting Items for a Plot
This menu shows all the different types of items that can be included in a plot. This menu is slightly different in that there is a small blank box just to the right of each item. To pick an item, you must pick the small blank box instead of picking the words. Notice how a number of the items have already been selected. These are the most typical selections for an "Artwork Plot". Although not typically required for an Artwork Plot, we will pick the Board item by picking the blank box to the right of the word "Board". This will include the Board Outline on the plot in

addition to the items already selected. Now pick the word "PROCEED" to move on to the next menu.

Selecting Transformation Options

This menu is more commonly referred to as a "Form". The text highlighted in Aqua shows the "fields" in the form that can be modified. The first field will have a flashing "text cursor" which indicates the field that is currently active for modification. To modify any field, the cursor must be moved to the required field, and then a new value can be entered from the keyboard. The arrow keys on the Numeric key pad or the mouse cursor are used for positioning the text cursor. The Down arrow goes to the next field; the Up arrow goes to the previous field, the Left arrow moves the cursor to the left within a field; and the Right arrow moves the cursor to the right within a field. Play with the arrow keys for a while to observe their effects, but do not type anything into any of the fields. For now all that we will do is to change the Plot Scaling Ratio from "1 to 1" to "2 to 1". To do this, first position the text cursor with the arrow keys over the "1" in the first field on the line with the words "Plot Scaling Ratio:"--then type the number "2". If you made a mistake, move the cursor back over the character you just typed with backspace and press the "Del" key on the numeric key pad and then type the correct key. We will accept all the other options "as is."

Now move the mouse cursor to the bottom of the screen and pick "PROCEED".

Selecting Processing Options

The new menu now appears. The first option "Add Selections to Batch File" will already be selected. Leave this selected and pick the option "Proceed with Current Selections", followed by the option "PROCEED".

The screen will go blank and the "Working" message will appear on the left hand side of the screen. The number displayed next to the word "Working" shows the progress time to complete the plot. It will start off at 99 and count down to 1. At about 40 remaining, the Matrix printer should start to print the plot you selected. The time to plot will depend upon your printer and the size of the plot. When the plot is done, the original menu will be displayed, and you are now ready to do another plot.

## 6.2. ARTWORK, LAYER 2

Repeat all the steps you went through for the Artwork, Layer 1, but where you select Layer "1" in the step for "Selecting the Plot Type," instead, you select the box with the number "2" in the menu.

## 6.3 ASSEMBLY DRAWING

Repeat the steps for Artwork, Layer 1 but stop as soon as you get to the "Select Plot Type" menu. Instead of picking "Artwork Plot," pick "Assy Dwg-Top Side" for an Assembly drawing of the top side of the board. Note that the level numbers now appear with two levels already selected. Level "1" is selected to show that you are plotting the Top side of the board and Level "27" is selected since this is the special "documentation" level reserved for the top side Assembly Drawing. It is not necessary to change the levels selected, so continue on by picking "PROCEED". The next menu is the "Select Items" menu, and it will show all the items normally selected for an Assembly Drawing, so continue on by just picking "PROCEED". The remaining steps are the same as for Artwork, Layer 1.

## 6.4 SOLDER MASK, LAYER 2

Repeat the steps for "Assembly Drawing" but pick "Solder Mask" instead. As with the Assembly Drawing, the levels will appear already selected. Level "2" will be selected for the Bottom side of the board and Level "28" will be selected for the Solder Mask documentation level. All pad sizes are plotted oversize. If required, the oversize amount can be adjusted on the Transformation Options Screen. Do not change the selected levels, but continue by picking "PROCEED". The rest of the steps are the same as for Assembly Drawing.

## 6.5 DRILL DRAWING

Repeat the steps for Artwork, Level 1 but stop as soon as you get the "Select Plot Type" menu. Instead of picking "Artwork Plot," pick "Drill Drawing". When the level numbers are displayed, Level "24" will be selected as the documentation level for Drill Drawing. It is necessary to pick one of the physical levels of the board, i.e., Level 1 or 2 for this

board. All the drill holes drilled through the level selected will be plotted. For now, pick Level "1" and then pick "PROCEED" to continue. As with other plots, the items required are already selected on the "Select Items" menu so continue on through the remaining steps which will be the same as the other plots.

The drill drawing plot will show a different symbol for each different drill size at each drill hole location. Before the plot, a short report is printed on the printer listing the symbol assigned to each drill size. The symbols can be changed if desired by using the DEFAULTS, MISC option at the opening menu. Refer to the Defaults section in Chapter 11 of the Reference Manual for more details.

## 6.6 PHOTOPLOTTER ARTWORK

The previous section of this chapter discussed the use of the Matrix Printer as a CAM Output device. In this section we will discuss the use of a photoplotter as the CAM device.

PADS-2000 outputs to both the Mechanical 24 aperture Gerber photoplotter and Laser photoplotters. The target photoplotter must be set during PINSTALL. If the target photoplotter changes with time, the Hardware setup of PINSTALL must be reset.

Have XXX12 (the completed board) displayed. We shall create a Layer One Artwork file and after having created the file, we will check it (validate) by creating an exact copy of the photoplotter file using the Matrix printer. (In this fashion, you can check your photoplotter file before ever going to the photoplotter.

Select CAM. Specify the CAM output directory as XXX12. You now have all the XXX CAM files in the same directory, such that after post processing you can delete all the files for that board, by deleting the directory.

Let's first make a Gerber file of Layer One Artwork.

Select DIRECT, select PHOTOPLOTTER, select ARTWORK plot, LAYER 1. Next, accept all items identified under L1 (pads, vias, tracks, etc.) by selecting PROCEED. Next, accept all the

Transformation options by selecting PROCEED. Next, select "Proceed With Current Selections," then PROCEED.

The file is created IF all apertures required for the tracks and pads are available with the Aperture Table. If not, a message will be generated, informing you of the apertures not available. A typical report is as follows:

"Plot requires new aperture assignments"

Apertures used: -
No: 11, Width: 10, Hgt/ID: 0, Shape LINE
No: 12, Width: 12, Hgt/ID: 0, Shape LINE
No: 15, Width: 50, Hgt/ID: 0, Shape LINE
No: 16, Width: 70, Hgt/ID: 0, Shape LINE
No: 17, Width: 55, Hgt/ID: 0, Shape RND
No: 26, Width: 200, Hgt/ID: 0, Shape ODD

Apertures required:-
Width: 7, Hgt/ID: 0, Shape LINE
Width: 60, Hgt/ID: 0, Shape SQR
Width: 60, Hgt/ID: 0, Shape RND

According to this report, there is a 7 mil line in the board, but no 7 mil aperture in the photoplotter table. Additionally, there is a 60 mil Round and 60 mil Square pad requirement but these apertures are NOT in the Aperture Table.

You should have a similar message while generating your Gerber file. If so, you must change your Aperture Table to include the missing apertures. This is achieved within the Defaults menu selection in CAM.

Select EXIT, select DEFAULTS, then PHO-PLOT. Your current Aperture Table is displayed, and you must add the missing apertures. Either select an unassigned D code or a D code that is assigned but not used on XXX12 and add the missing apertures. You will be prompted to update the Master file [DEFAULTS.PAD]. A response of "Yes" will update the Master file, thus all future aperture tables will contain these new apertures. A response of "No" will update this Aperture Table only, within the \CAM\XXX12 directory.

NOTE: This information is stored in binary form in the file BATCH.DAT, contained within the user specified subdirectory.

NOTE: At the end of your training, you should consider what apertures you will need in your firm and then change the Master Aperture Table accordingly. In this fashion, you will not need to set each time for a new job.

Once set, go back and create your Gerber file. Check the Aperture Table printout and be sure to include it with your Gerber files, when sent to be photoplotted.

Now that the file has been created, it exists in the CAM directory XXX12 with the name ART01; Layer 2 Artwork would be ART02, etc. Let's plot it with the Matrix Printer for visual inspection before sending to the photoplotter.

Select CAM, select Directory XXX12. Select VERIFY. In response to the request for Gerber file name, respond ART01.

NOTE: The Gerber file names are assigned automatically at the time of the file creation. See Section 11.8, CAM OUTPUT FILES, of Volume II, the PADS-2000 Reference Manual, for the Assignment logic.

Select Matrix Printer since we shall verify on the Matrix Printer. Accept the Transformation options with PROCEED. After a short delay, the printer will print out an exact copy of what the Gerber file will be.

## Chapter 7

# TUTORIAL 6 - ASCII FILES, THE PARTS LIBRARY, AND DESIGN-ON-THE-FLY

(240 minutes)
In this Tutorial we will cover the items involved in starting up a new board. These are the ASCII Parts List, the ASCII Connection or Net List, the preparation of System Library Part Type items and the preparation and maintenance of Part Decals. As an alternate method of entering data we shall cover Designing-On-the-Fly.

## 7.1 ASCII INPUT FILES

To enter a new board into the system, two lists are required:

1) a Parts List in ASCII
2) a Connection List in ASCII

These may come from a Logic Capture system such as PADS-LOGIC or you may generate them using the ASCII Editor (PEDIT) in the software. You are to do the latter on the training board schematic shown in Figure 2.3, Chapter 2.

### 7.1.1 PARTS LIST

From the Parts List provided you in Figure 2.4, generate an input Parts List. Read Section 13.3.2, Parts List, of Volume II, the User Reference Manual, regarding the Parts List format. Create the Part List using paragraph 13.3.2 as your guide. In the preparation of the Parts List, pre-locate the Edge Connector as per the Design Criteria, Figure 2.6 Also, remember to add Moires and Mounting Holes to the Parts List (they are NOT on the schematic).

When the Parts List is created, use the System Editor PEDIT to enter the Parts List into the subdirectory FILES. See Chapter 13, Section 13.6, the Editor, of Volume II, User Reference Manual, for a description of the use of the System Editor (PEDIT). Print out a copy and compare it to the copy used to originate the file you have used in your training sessions. This is shown in Figure 7.1, ASCII Parts List.

```
*PADS2000
*PART*
*STANDARD
U1      74139
U2      7404
U3      7402
U4      7474
U5      7432
U6      7404
U7      7440
U8      7420
U9      7400
U{12-21}      6167
C1      CL25,10UF
C2      CL25,10UF
C{3-22}       DCAP1,.01UF
R1      R1/8W,1K
R2      R1/8W,1K
Y1      XTAL1
P1      CON\26P\ED 700 -300
MH1     MTHOLE1
MH2     MTHOLE2
M{1-3}    MOIRE
*END*
```

*Figure 7.1 - ASCII PARTS LIST (PART.ASC)*

NOTE: The use of the header "STANDARD" will extract the Ground and Power connection for those Part Types in the Library that have standard VCC and Ground Pins identified.

Notice the shorthand format for C3 through C22, U12 through U21 and Moire M1 through M3.

NOTE: The format of U1 74139@DIP16 is also suitable. The @DIP16 overrides the Decal as assigned to 74139, in the PADS Library. USE WITH CAUTION!

### 7.1.2 CONNECTION LIST

After finishing the Parts List, you are to prepare a Connection List from the schematic of Figure 2.1 Read Chapter 13, Section 13.3.3, Connection List of Volume II, User Reference Manual, as background for the preparation of the Connection List.

Follow the format **exactly**! Enter your Connection List into the subdirectory of FILES using the System Editor (PEDIT).

After entering the Connection List into the subdirectory, make a printout and double check the listing against the schematic.

### 7.1.3 READING ASCII IN FILES

Upon completion of the Part List file and the Connection list file, you are ready to read them into the system. You should read them into the system memory together with the Board Outline. File XXX1 from Tutorial 2 contains the Board Outline. Using IN/OUT, read it into memory and display it at the screen. Then, using IN/OUT, ASCII IN, read in the Parts List you just prepared. (Always read in the Parts List before the Connection List.)

As the Parts List is read in through the ASCII IN part, a number of checks are made on the data. If discrepancies are found, an error file or printout is created. Above a specific number of errors, the read in process in aborted until the errors are cleared. A number of discrepancies may result from reading your Parts List file. In all probability, these result from not having a Part Type called for in your Parts List in the System Library. Another potential error is not having the Part Decal associated with the Part Type, in the System Library. You might also have a Parts List format discrepancy. In any event, when a discrepancy occurs, read the error listing carefully, analyze it, together with the Parts List, and the proper course of action should be clear.

Edit the Parts List, using the Editor (PEDIT) until it can be read into memory and merged with the Board Outline, without generating error messages. After being read into memory, the Board Outline and Parts can be viewed at the screen.

Make a file on the hard disk (JOB OUT) of the Part List and Board Outline, for safety reasons. With this file in Memory, read in the ASCII Connection List you prepared from the schematic.

The Connection List is checked for errors, while being read into memory and merged with the Parts List. Again, you may have some error indications. Carefully analyze them, checking against the schematic. Correct the Connection List using the Editor, until it can be merged with the Parts List and Board Outline without discrepancies. Once merged, it is identical to File XXX2 you used in Tutorial 2 to start the training board.

NOTE: Some of the most frequently encountered errors are:

1) Not having the Parts already in memory before reading in the connection list.
2) Creating the Parts List using the REF DES and Part Decal rather than REF DES and Part Type.
3) Not having the board outline and board origin setup at the time the Part List is read in (not an actual error but also not recommended).
4) Creating the Power and Ground connection on the Schematic and also creating the Power and Ground connections for parts whose Part Types have the Power and Ground pins identified.

With the Board Outline, Parts List and Connection List read in successfully, make a file and record it in your Job Log as "BO, Parts and Connections."

## 7.2 ASCII IN - ASCII OUT

PADS-2000 is open architectural in that it can be interfaced to other systems using the ASCII OUT Routine of IN/OUT. Similarly, PADS-2000 can accept inputs from other systems through its ASCII IN routine. In both above instances, the formats of the other system must be compatible with the PADS-2000 ASCII.

## ASCII

To create a PADS-2000 ASCII Out file, have a board loaded and select ASCII OUT (F4). A Pop-up Window will appear as follows:

| | | |
|---|---|---|
| PCB | MATRIX | TEXT |
| LINES | DECAL | PTYPE |
| PART | CONN | TRACK |
| ALL | | |

In addition the menu changes to:

**1 Select** **2 Go**

**10 Exit**

As seen from the pop-up window, you can produce separately, nine different sections of the database in ASCII, by SELECT (F1), together with picking the item, then GO (F2). Alternatively, if the entire database is desired, select ALL. After either selection, choose GO (F2) and respond with the ASCII File Name. The file will be created using this name together with the extension .ASC.

NOTE: The PCB section of the ASCII OUT Window, called PCB is the database SETUP file.

Let's make and examine the ASCII OUT of XXX12.

Have XXX12 displayed. Select ASCII Out. Place the cursor over ALL in the ASCII OUT window and select SELECT (F1). Select GO (F2) and enter the name of the file XXX12. You do NOT want to correct UNITS. A file is created. Let's examine it.

Select ALT 9 and give the file name of XXX12.ASC and the file will be displayed. Let's examine:

The first part is the Setup file.
The second part defines the Matrix.
The next part describes Text.
The next part defines the Board Outline and all 2-D Lines on the board.
The next part defines the Part Decals.
The next part defines the Part Types in the design.

The next part defines the Parts List.
The next part defines the Tracks/Connections.

HINTS:

You may find some things easier to edit by using a Text Editor rather than using a Graphics Editor. If so, make an ASCII Out using ALL, do the editing, then read back in using ASCII In.

Suggestion: Database corruption may occur from time to time, as a result of power failure, memory failure, software bugs, etc. You may be able to correct the corruption by making an ASCII Out file (printed) and inspect it. If you are able to locate the problem, use the Editor to make the correction. If you cannot, try to read the ASCII Out file back into the system using ASCII IN. In most cases, the Error Checking feature of ASCII In will identify the error, thus allowing you to correct it.

## 7.3 DESIGNING-ON-THE-FLY

An alternate method to the use of the Parts and Connection Lists or a Schematic in the creation of the database for a board is the technique of Designing-On-The-Fly at the CRT. The parts and connections are entered at the CRT on a part-by-part, connection-by-connection OR track-by-track basis.

For our self-teaching example we shall use Sheet One of the Training Schematic, Fig. 2.3a, together with the file TONTHEFL.JOB that has been prepared for your use.

Using IN/OUT, JOB IN, call in file TONTHEFL.JOB. This file contains the Board Outline together with the connector P1 already placed in its appropriate location.

Your training exercise is to design Sheet One (Fig. 2.3a) by entering the Parts and Connections into the file TOTHEFL.JOB, using the subroutine of IN/OUT called (F8) ONTHEFLY.

Select (F8), ONTHEFLY of the IN/OUT Routine. The menu changes to:

**1 Add Conn 2 Del Conn 3 Rename Net 4 Disconn 5 Add Part**
**6 Del Part 7 Ren Part 8 Mod Rut 9 Add Rut 10 Exit**

## 7.3.1 ADD PARTS AND CONNECTIONS

Using ADDPART (F5), you will add the parts contained on Sheet One of the Training Schematic, Fig. 2.3a.

Select ADDPART (F5) using KEYBOARD I/O (F2), add U2 (U2 part name is 7404 - see parts list, Fig. 2.4). Rotate and place in the correct location, referenced to P1, as suggested by Sheet One of the Training Schematic.

In response to the prompt:

**COMPONENT ALPHA PREFIX FOR NEW PART TYPE>**

respond U for IC, R for Resistor, etc.

You will notice the first IC you add will be designated as U1, the second as U2, etc. This is the Automatic REF DES system assigning REF DES. Do not be concerned that they don't match the schematic at this time. Later you will change them so they do match.

Now ADD and PLACE the following parts, using the schematic as a rough placement guide and the Parts List, Figure 2.4 to select the correct Part Type for each part.

| | |
|---|---|
| U3 | 7402 |
| U4 | 7474 |
| U5 | 7432 |
| U6 | 7404 |
| R1 | R1/8W |
| R2 | R1/8W |
| Y1 | XTAL1 |

At this time, with all the components placed, use RENAME PARTS (F7) to make the Ref Des the same as Sheet One of the Training Schematic. Since the PADS Error Checking feature prohibits using the same part name in multiple applications on the same board, you may

need to use an out of sequence name for some of the ICs, then rename them to their proper name.

Using ADD RUT (F9), add the connections on Sheet One, Fig 2.3a. Power and Ground are 0.050" tracks, Signal connections are 0.012" tracks.

Set the Real Width to 1 (in order to see the tracks with a width).

First, add the Signals at 12 mils. Use W12 <CR> and observe the Width is WID 12.

Change the width to 50 mils and add Power and Ground. Rename Power and Ground nets to PWR and GND using REN NET (F3).

Relocate parts as necessary by the connection pattern.

Ignore connections to Off-Sheet Page Symbols, for this exercise.

HINT: Select a pin on a 2-layer Edge Finger connector using the keyboard, i.e., P1.4, etc.

HINT: End the track with COMPLETE (F9)

Run Space Check, Run Reports, Net List, Run Reports Parts List, check the Part List and Connections List against the schematic.

## 7.4 LIBRARY PARTS

In the remainder of this Chapter we shall learn the basics of creating the two entries which comprise a part and storing them in the Parts Library. The software already contains a large number of parts including parts to do the Training board. Nonetheless, you may need to create new parts for the first board you design. Save the parts in the Parts Library and as you design more boards, your library will become more and more complete.

When you are creating a new part you must always remember that there are two separate entries you will need to make, in order to create a part. These entries are as follows:

(1) The Part Decal - a graphics entry mode at the CRT, using the CREATE Routine. The Part Decal contains the Component Outline and Pads. The Part Decal is the footprint you see at the CRT and is plotted. You will make a Part Decal in Section 7.4.4.

(2) The second entry of creating a part is the entry to create the Part Type electrical information describing the part.

NOTE: If you have both the PADS-LOGIC Schematic software and PADS-2000, the Part Type data associated with a part is created in the PADS-LOGIC Library Routine used to create Parts. Data created in this fashion is available to both the LOGIC and PCB programs with their integrated library.

If you do NOT use the PADS-LOGIC software, you will need to create the Part Type data for a part, using either the PLIB Library Text Editor provided with PADS-2000 or the CREATE, PART TYPE Routine in PADS-2000. In this manual, we shall assume you do NOT use the PADS-LOGIC software to create Part Type data and, therefore, you will use PLIB to create the data. You will also practice with CREATE, PART TYPE.

Each part has a unique Part Type description, unique to that part. Many parts however, can have the same Part Decal. All 14 pin IC's for example have unique part types but all may use the same 14 pin Part Decal.

A Part Decal is related to the Part Type by the part decal name which acts as a "pointer". This pointer is contained within the Part Type information.

When a Parts List is created, it must contain the Reference Designation and the Part Type name (not the Part Decal name). When the Parts List is read into the system, the Part Type is first retrieved from the Part Type Library. Following this, the Pointer in the Part Type is used to retrieve the associated Part Decal. At this time the Part Type and Part Decal are brought onto the board database and the Part Decal is displayed at the CRT.

NOTE: Using the Part Decal Name, U1 DIP14 in the Part List, is the most common error; take care to use the Part Type Name, U1 7404.

## 7.4.1 PART TYPES (Using PLIB)

There are two ways to create Part Type data: (1) PLIB (Section 7.4.1 and (2) the routine, CREATE, PART TYPE (section 7.4.2)

The PADS-2000 software contains a System Library in which is located the data for the parts used in the Parts List. As you have seen, while reading the Parts List into the system using ASCII IN (or using ECO or Design-on-the-Fly), a part identified on the Parts List must be available in the System Library, using as its name, the name associated with its Reference Designation in the Parts List.

Although a large number of parts have been included in the System Library, it is absolutely essential that you be capable of adding Part Type data to the System Library. Accordingly, at this point, go to Chapter 12, System Parts Library, Volume II, User Reference Manual, and thoroughly study the concept and format for Part Types in the System Library.

Following this, read Appendix A-4, to provide you with a summary overview of the Library and PADS System integration.

Having studied Part Type Library content, you will appreciate that the complete format for a Part Type varies between two extremes in terms of data content. On the one hand, the minimum amount of data for a Part Type can be contained in only one line of data and, in fact, only two data entries on the one line, i.e., the Part Type Name and the Part Decal name. In fact, if the Part does not have a standard Power and Ground connection and you have no interest in Gate and Pin Swapping, and you are not interested in cost data, you might consider using only the minimum, two items of data content format. Certainly, if the Part has Standard Power and Ground pins, you should always take advantage of this feature to generate your Power and Ground Lists.

If you desire to build a complete Parts Library, you will want to use the complete data format.

In the training example, we will use the latter, the complete data format.

Let's say that you want to add a new IC to your Parts file. This IC is to be 7474, a dual Flip Flop. (Refer to Texas Instruments Data Manual for the vendor data).

Using the format content in Section 12.5.2, Part Type File Format, Chapter 12, Volume II, User Reference Manual, you prepare the data for the IC.

Note that 7474 uses a 14 pin package, therefore, the Part Decal is DIP14; it has 2 gates, and 2 pins with Signal names (GND and +5V). Try your hand at the format and compare your data against the format as shown in Figure 7.2, Part Type Data for a 7474 IC.

```
\74  DIP14: DIP14\SO  TTL  6 2 2 0 0 0
@X4  X4ALS  X4HC  X4S  X4HCT
PART DESC:  DUAL D-TYPE POSITIVE EDGE TRIGGERED FLIP
FLOP
$:
PART NUMBER:  TBD
MFG. #1: TEXAS INSTRUMENTS, INC.
MFG. #2:
G:\74  1  6
4.0.L.\S  3.0.L.CLK  2.0.L.D
1.0.L.\R  5.0.S.Q  6.0.S.\Q
G:\74  1  6
10.0.L.\S  11.0.L.CLK  12.0.L.D
13.0.L.\R  9.0.S.Q  8.0.S.\Q
SIGPIN  7  50  GND
SIGPIN 14  50  +5V
```

*Figure 7.2 - PART TYPE DATA FOR 7474*

Let's also examine the Part Type data for a 7474 as contained in PLIB.

1. Exit PADS-2000 and select PLIB from the Shell.
2. Select the Standard Library: S
3. Select Part Type: P
4. Select Find: F
5. Enter Item: 7474

Data is displayed in Edit Window. Be sure you understand the significance of each entry. After reviewing this data, select "B" for Bye, once again "B" for Bye.

Now that you understand the format, let's create a Part Type 74744 in the User Library using PLIB.

Be in PLIB, select User lib, Part type and Edit. Enter the data for your 74744 using the 7474 specifications and SAVE. Note that if there are errors, the line containing the error will be RED, and the Part cannot be saved until the errors have been corrected.

How about discrete parts? Resistors for example?

A major problem arises with discrete parts: the large number of parts involved (approximately 15000 Resistors, for example).

To simplify this problem, yet permit you to include detailed Part identification we suggest the following:

1) Categorize Discrete parts into families as per the decal each family will require. Resistors, for example, would have a different part decal for 1/8 or 1/4w, 1/2w, 1 w, etc.

2) Create a Part Type for each family and use the family Part Decal name for the Part Type and SAVE.

3) Create the Part Decal for the family using the same name as the Part Family and SAVE.

4) When you call up the Part Type and its associated Part Decal with ECO, On-The-Fly or Parts List, you may add additional data called "tags" (up to a total of 16 characters) such as 1000 OHMS, to the part type name so as to uniquely identify the part. The entire data including the tags, will be carried forward to the Parts List.

ALTERNATES

The following Part Type entry will illustrate the concept of alternate Part Decals, if you plan to use Alternate Part Decals.

R1/4W R1/4W: R1/4WA: R1/4WB RES 8 1 0 0 0 0

where:

The first R1/4W is the Part Type name for the family of 1/4 watt resistors.

The second R1/4W is the name of the Primary Part Decal.

The following R1/4WA is an alternate Part Decal (an on-end Decal).

The following R1/4WB is an alternate Part Decal(45º decal).

NOTE: The various Part Decals are separated in the Part Type header by a colon, thus :

When the Part Type is called up on the board, the PLACEMENT routine of MOVE, ALTERNATE will allow the designer to select between the three Part Decals.

### 7.4.2 Part Type (Using PADS-2000 CREATE)

PADS-2000 provides two different methods of creating/modifying Part Type data. The first is using PLIB, the Part Library Editor as described in 7.4.1 preceding. This method is provided for those persons that prefer a "free form" format as used by the Part Library Editor.

Other persons may prefer a "Question and Answer" format. This format is contained in the CREATE, PART TYPE (F9) Subroutine. We shall use this feature in this section.

Prior to this practice, go to Section 12.6 of Volume II, The PADS-2000 Reference Manual and review the details of the CREATE, PART TYPE feature.

The PART TYPE Subroutine is located in the Top Menu of CREATE as PART TYPE (F9). When PART TYPE (F9) is selected, the following menu appears:

**1 New Part  2 New Conn  F3 Modify**

**10 Exit**

NEW PART (F1) is used to create a new Part Type, other than a Connector.

NEW CONN (F2) is used to create a Part Type for a Connector.

MODIFY (F3) is used to modify an existing Part or Connector.

### 7.4.2.1 Creation of a Multi-Gated (74LS02 IC) Component

In this section, you will create a TTL Integrated Circuit Part Type. To give you a preview of what you will be creating, go to the CREATE routine and select PART TYPE (F9), then MODIFY (F3). In response to the request for the Part Type, respond 74LS02. Select the 74LS02 from the pop up menu.

There are two parts to making a Part Type. The first is Part information and the second is the Gate Information. You will note these under the PART TYPE menu as PART INFO (F1) and GATE INFO (F2).

Select PART INFO (F1) and the Part information for the 74LS02 will be displayed.

You know from your experience that a 74LS02 is a 2-Input Positive NOR device containing four Gates. When looking at the Gate information, GATE INFO (F2), at the CRT, you see four gate names. The reason for this is that the Primary Gating information is shown in the four gates on the left, and the DeMorgan Equivalent Gates information is shown on the right.

Pay attention to the Pin Numbers and the Pin Type and Swap information associated with the Pins. Study this information carefully since you will be creating the information on your own later on for a different IC.

Next, examine the Part information; PART INFO (F1), for the 74LS02. Study the entries for both Part and Gate info and based on the description for the Part information in Chapter 12, paragraph 12.6, Volume II, the PADS-2000 Reference Manual, try to determine why the entries are as shown.

When you are satisfied that you have a reasonable understanding of the contents of the PART and GATE INFO sections of a Part, EXIT and bring up PART TYPE (F9), then NEW PART (F1). To the prompt:

**Name of the New Part Type**

use the name \022 rather than \02. We shall recreate the info for a 74LS02. The reason for using this name is to insure that you do not overwrite the "good contents" of the 74LS02 as contained in the Library.

What you will actually be making is a part we will call 74LS022--a non-existent part but based on a 74LS02--that you can delete from the Library after you have learned the fundamentals of creating an IC part.

The first question that we will address is what to call the Part. We could indeed call it simply 74LS022. That is perfectly legitimate and if it were one of a kind, rather than a member of a family, that's what you would call it. You will recall that the 74LS02 is a member of a large TTL family, 74LS02, 54LS02, etc., and we do not want to create a part for each of the members of the family, and thus we use the concept of Suffixes and Prefixes. If you don't recall the exact definition of Suffix and Prefix, go to Volume II, Section 12.6 and read the paragraphs on Suffix and Prefix.

After having read this section, you recognize that we will call the Suffix of the Part \022 and thus we respond to the Part Type Name as \022.

The Part Info Text screen will be presented at the CRT. Notice that the Suffix has already been entered as a \022.

Prefix
Here is where we identify the entire family for the 022. Go to your IC Data Book for a 7402 and determine the number of different 022's supplied by the vendor and enter the Prefixes in accordance with the instructions for Prefixes in Volume II, Section 12.6. Upon completion, your Prefix entry should look approximately as follows:

@X4 X4ALS X4LS X4S 54L X4HCT

Logic Family
Since this is a TTL Part, enter TTL. The significance of this entry amongst other things is that in accordance with the data contained in the PKGTYP.DAT file, anytime you use a TTL part, the Reference Designation will start with a U.

PCB Decal and Alternates
This is a 14-pin DIP Part and the PCB Decal is called DIP14 therefore, enter DIP14.

Number of Part Attributes
The maximum amount of data for a Part is ten lines of data. Let's enter 5 and 5 lines for Attribute data entry will be displayed.

On the first line, enter the Parts Description.

On the second line, enter the Unit Cost.

The third Attribute Line is for Part Type or Company Part Number for Digital parts and VALUE for Discretes. You would enter the Part Type Number, i.e., \022.

The fourth and fifth Attribute Lines are used to indicate the Manufacturer. Alternately, on a Discrete Part the fifth line can be used to indicate Tolerance, i.e., 1%, 5%, etc. To use, enter TOLERANCE: ??? When the part is brought on to the Schematic, EDIT TEXT can be used to change the "???" to the Tolerance Value.

NOTE: If you are not using PADS-LOGIC, ignore the Value ??? and Tolerance ??? data.

Non-Numeric Pin Number
14-Pin DIPS use Numeric Pin Numbers, therefore, leave this entry as N (No) for the Default.

Number of Gates
In the data entry lines of Gate data, enter 4 and four lines for Gate data will appear on the screen used for GATE INFO, next..

Number of Signal Pins
The 74LS022 has two Standard signal pins, i.e., GND and +5V, therefore, enter **2**.

In the first of two data entry lines, enter information for Ground, which would read Pin Number 7, Signal Name is GND and 50 Mil Track Width. On the second line, enter Pin Number 14, Signal Name is +5V and 50 Mil Track Width.

By entering the "standard" Power and Ground Pin data into the IC Library Part, the Power and Ground Connection to the Net, for this part is automatically created when you add the part to the Schematic or Board. Connecting the Standard Power and Ground Pins of the part to a Power and Ground Symbol is not necessary!

Next, you will complete the Gate Info. Select COMPLETE (F9) to complete the part info. Select GATE INFO (F2).

Notice that the system has automatically created the four Gate Entry Lines for the four Gates called out in the PART INFO (F1). If you examine these Gates carefully, you will notice that the Pin Numbers are absent, as are the Pin Type and Swappability information. You must, at this time, complete this information by entering data for each gate.

SW
SW = swappability between the four gates where 0 = Not Swappable and 1 = the Gates within a particular family of IC's (i.e., U1) are swappable. Gate A of U1 can be swapped with Gate B of U1 and Gate A of U1 can be swapped with any gate of any other \022 part on the board. Enter SW = 1 since Gates are swappable.

PINS
Number of Pins in the Gate. There are 3 pins. Enter 3. This creates an Entry Line for entering the Pin Info for the three pins.

PADS-LOGIC Gate Decal and Alternate
This is NOT required if you are NOT using PADS-LOGIC as your Logic Capture system. If using PADS-LOGIC, enter the name of the Schematic Gate Decal, a space and its DeMorgan Equivalent.

The Decal for each Gate will be a NOR Gate, therefore, enter NOR2, and since there is a DeMorgan Equivalent, provide this Decal information also, by entering the DeMorgan Equivalent Decal Name which is NORDM, with the Primary Gate Decal and the DeMorgan

Equivalent separated by a space. Enter this information for all four gates.

Enter: NOR2 NORDM

Second Line

Pin #: Enter pin numbers in Gate A. Enter: **1**

T: = Pin Type
S = Source Pin
L = Load Pin
B = Bi-directional pin
C = Open Collector
O = Or-tieable
T = Tri-state

Enter: L

Complete for all Gates and all Pins. Upon the completion of both the Part and Gate information, you should save the Part. Do this by first selecting COMPLETE (F9), then SAVE (F9). Watch for the "PART SAVED" message. If in doubt, save again!

When you are satisfied that you understand how to create multi-gated IC's, delete this \022 Part from your library, inasmuch as it is a non-part. (Use PLIB to delete the Part--see Chapter 12, Section 12.5 PLIB, on the use of PLIB.)

### 7.4.2.2 Creation of a Discrete Part

In this section of the tutorial, we will create a simple discrete component, specifically, a Resistor.

In general, the creation of a Discrete Part is identical to a multi-gated IC except for certain specifics which will be brought out as you create the Resistor.

Go to PART TYPE and MODIFY (F3). Call up a R1/8W and examine the Part Info and Gate Info for an existing Discrete Part. Study both the PART INFO and GATE INFO.

We will call the new Resistor we are going to make, R1/4WW to distinguish it from the real R1/4W Part that is in the Library.

Before we start making the Part, we want to insure that a correct Gate Decal exists; if not, we will need to create one.

Find out whether a Gate Decal exists. Go to the routine CREATE, PART DECAL, MODIFY and in response to the request for the Name, enter R*. We are making an educated guess that the Decal for a Resistor is called Resistor or something similar and, therefore, by using R with the wild card *, we are asking for a listing of all Gate Decals starting with R to be displayed. The Decal exists and, therefore, we will first work on Part Info items.

Select PART INFO (F1) of NEW PART and in response to the prompt:

**Enter Part Type >**

respond R1/4WW. Select PART INFO (F1) and proceed to complete the Part Info data.

Part Type
This is already entered for you as R1/4WW.

Prefix
We do not need a Prefix.

Logic Family
Enter RES. The significance of entering RES is that any time this part is used, the Reference Designation will be started with an "R." This is determined by the PKGTYP.DAT file.

PCB Decal and Alternate
In PADS, the Decal for a 1/4W Resistor is also called R1/4W, therefore, enter R1/4W.

Part Attributes
Part Attributes can be used to display and plot other information in PADS-LOGIC. Up to ten lines are possible. Enter 5. For example, a Part Attribute can be used to display and plot the ohmage and Tolerance of the Resistor. Since we will want to do this with a Resistor, enter VALUE: ??? on the Third Attribute Line.

Attribute Lines have a special significance in the instance of Discrete Parts, such as Resistors or Capacitors, wherein you may wish to display at the CRT, plot on the drawing and list on the Parts List, certain values such as OHMS, Farads, etc. Specifically, Attribute Line #3 can be used to assign these values. When so used, in Attribute Line #3, enter the word "Value" followed by three question marks, thus VALUE: ???.

On the First Attribute Line, enter:

**Part Desc: Resistor, 100 mil centers**

On the Second Line, enter:

**Part Number: TBD**

On the Third Line, enter:

**VALUE ???**

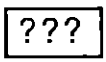
On the Fourth Line, enter:

**MFG #1**

On the Fifth Line, enter:

**TOLERANCE: 5%**

When this particular part is brought up and added to the Schematic on the CRT using ADD PART. You will note on the Gate Decal for the Part, that the VALUE and TOLERANCE Displayed Attribute Line will show:




You may set this Value as desired using the EDIT TEXT routine in PADS-LOGIC.

NOTE: The above editing of VALUE and TOLERANCE is NOT done in the PART INFO creation phase, but rather while you are creating the schematic in PADS-LOGIC. The above information is provided at this point simply to show how the features would be eventually used.

Number of Signal Pins

A common Resistor will NOT have Power and Ground, therefore, leave the Default of 0.

Number of Gates

There will be a single Gate or Decal, therefore, Enter **1**

Non-Numeric Pin Number
Resistors use Numeric Pin Numbers (although suppressed at the CRT and on the plot), therefore, leave the Default as No.

When complete, check your Text File and use COMPLETE (F9), then SAVE (F9) and check for the prompt:

**Part Saved**

Your will now complete the Gate Info. Select GATE INFO (F2).

Let's start on Gate A.

Swappability
You do not want Resistors swapped with other Resistors, therefore, leave as **0**.

Pins
In this instance, enter **2** and a second Text line will appear.

PADS-LOGIC
(Not required if Part is used only in PADS-PCB or PADS-2000).
Enter: **RESZ-H** and **RESZ-B** (for the Primary and Alternate schematic resistor symbols)

In Next Line:
Pin #
Enter: **1**

Type
Enter L for Load or U for Undefined.

Swappability
If you wish Pin 1 and Pin 2 on a particular Resistor to be swapped so as to shorten connection length, enter **1**. If you do not wish to swap pins, enter **0**.
Enter: **1**

Pin Name
Leave Blank

Repeat Pin 2 data for Pin 2.

Select F9 COMPLETE (F9) to finish the Gate Info and then use SAVE (F9).

7.4.2.3 Connector

In this section, we will create a new Connector.

Generally speaking, the creation of Input/Output Connectors is the same as the creation of Parts, except that in the process, one allows for the fact that when a Connector is brought on to the sheet of a Schematic, it is usually brought up at the Pin Level, rather than the entire Component Level and, secondly, the Connector Part is generally not shown on the sheet of the schematic, as would be a 74LS08 or other non-connector part. To do this, use a routine to create New Connectors, that is slightly different from New Parts.

In this exercise you will create a 10-pin Edge Finger Connector.

Regarding the Naming convention for Connectors, you should generally name a Connector in such a fashion as to use the Name to identify the type or characteristics of the Connector. In this particular case, we will call the connector CON\10P\ED. This nomenclature is for a Connector, 10 Pins, Edge Finger.

Be in CREATE, PART TYPE and select NEW CONN. NEW CONN is used exclusively to create Connector Parts.

Select NEW PART, then NEW CONN (F2). Enter the name CON/10P/ED.

The next level of menu is PART INFO (F1). Select PART INFO.

| | |
|---|---|
| Number of Connector Pins: | Enter **10** |
| Logic Family: | Enter **CON** |
| PCB Decal | Enter **CON\10P\ED** |
| Number of Part Attributes: | Enter **0** |
| Non-Numeric Pin Numbers: | Enter **(N)** |
| Line Required: | Enter **0** |

Use CREATE (F9), then SAVE (F9) to save the connector.

### 7.4.3 Modify Part Type

PART TYPE has a subroutine called MODIFY (F3). This routine is used to Modify both Parts and Connectors. Select MODIFY (F3) and call up a 7404 and attempt your hand at modifying the Part Type. Do not save or overwrite your good 7404.

### 7.4.4 PART DECALS

While, as we have seen above, the Part Type data for a part is primarily Part Description, Gating, Pinning and PWR/GND information, each Part Type has associated with it a Part Decal. The Part Decal is the pictorial or graphical "picture" of the part as used at the screen or plotted on a Drawing or Artwork.

The Part Type data identifies its Part Decal and this Part Decal must be contained in the Part Decal section of the System Parts Library. If it is not, then the Part Type data is incomplete, therefore, when the Part Type is called for on a Parts List, an error message will result when the Parts List is entered in the ASCII IN port.

With this in mind, let's get familiar with creating a Part Decal. To create a Part Decal, you must be in the CREATE routine of PADS-2000, and use the PRT DECAL (F5) subroutine.

Let's create the Part Decal DIP14 called for in the Part Type data for IC 7474 created previously. Part Decal DIP14 is already in the System Parts Library and we do not want to disturb it, as we practice creating our practice Part Decal. Accordingly, use the name TEST as the name of the Part Decal we are creating.

First, in order to create a Part Decal that can be used on either two or four layers, create it with a four layer database. Use SETUP (F2), MAXLAYER (F4) and set Max Routing Layer to 4. Set Old Level 1 to New Level 1 and Old Level 2 to New Level 4.

Next, using CREATE, PART DECAL, CREATE, TERMINAL (F3), ADD TERM (F1), enter the terminals of the Part Decal at their correct position relative to each other. Always set Grid to 100 when entering

DIP-type IC pad data. This will insure Pads are 100 mils apart. Add 14 terminals in their appropriate locations (refer to Figure 7.3). Using SETUP (F5), move the Part Origin over and coincident with Pin 1.

Next, using CREATE, PART DECAL, OUTLINE, 2-D LINES, draw the component Part Decal Outline. You will probably want to use a Grid of 25 while creating the Outline. Decide the size of the outline with respect to the Pad pattern. Do you want the outline inside or outside, on rows of pads, etc? We suggest the outline be inside the IC pads. Relocate the position of the Pad numbers, relative to the pads. Next, relocate the Part Name/Type relative to the Part Decal. Upon completion, your display at the screen should appear approximately as follows:

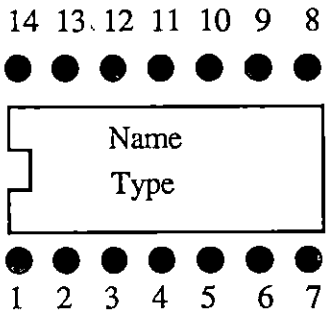



*Figure 7.3 - CREATION OF PART DECAL*

After a Part Decal is created, it is automatically assigned a Default Pad Assignment during the SAVE sequence, unless you made a Pad Assignment. Always set the pad assignments AFTER the pad/terminals have been created and placed on a Part Decal - NEVER BEFORE! Let's set the assignments in the training exercise as follows:

The routine for Pad Assignments, during the creating of a Part Decal, is exactly the same as when the Part Decal has been called from the Library and incorporated in a board, and the decision is made to change the Pad Assignment.

Carefully, read Section 4.2, Defining Pad Stack Size, Chapter 4, Volume II, User Reference Manual, for a description of setting

assignments on a pad. When you have an appreciation for the procedure, proceed to set the pads on the Part Decal you have created using TERMINALS (F3), PAD STACKS (F6). In the process, assume that the Part will always be used in multi-layer design. Therefore, you cannot ignore the "inner layers" question.

The following is provided to help you through the procedure.

The Default setting is as follows:
ALL pins to Round 60 mil, 37 mil Drill Hole
Top Layer 60 R 37
Inner Layers 80 R
Bottom 60 R

Top Layer 60 S 37
Inner Layers 70 R
Bottom 60 R

Your exercise is to change all 60 mil pads on Top and Bottom layers to 50 mils, and to set up for a Solder Mask (Layer 28) using ADD LVL (F6) and assigning 70 mil pads.

The CRT presentation should be as follows after the change:

Pad Stacks for Decal:

| | | | | |
|---|---|---|---|---|
| ALL | T | 50 | R | 37 |
| | I | 80 | R | |
| | B | 50 | R | |
| | 28 | 70 | R | |
| 1 | T | 50 | S | 37 |
| | I | 80 | R | |
| | B | 50 | R | |
| | 28 | 70 | R | |

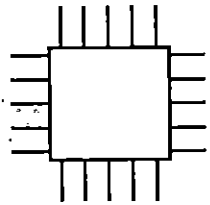
Repaint the CRT with Numeric Key 1, and visually check that the pad shapes and sizes are correct. Note that Pin 1, Layer 1 is a square.

(Study Section 4.2 of Volume II, the Reference Manual, for additional Pad Stack Assignments such as SMD parts, Connectors, etc.)

In studying the above example, you no doubt have determined that the concept to follow in setting pad stacks is to first set ALL pad stacks to a value that is correct for the majority of the pads--then following this, set the exception to the majority, as appropriate. In the example above, we would read the pad stack information in this Part Decal as follows:

Set all 14 pads to 50 mil Round Top and Bottom Layers, set inner layers to 80 mil Round, set Layer 28 (Solder Mask) to 70 mil Round and use a 37 mil Drill--except for Pin 1. Set Pin 1 to 50 mil Square on 50 mil Round on Bottom Layer, 80 mil Round inner Layers, 70 mil Round Layer 28 and use a 37 mil Drill.

Let's see how to use this on an IO pad SMD whose footprint is as follows:



You would proceed as follows:

Set all the pads to the same Size and Orientation.

Next, set the Vertically Oriented pads using an Orientation of 90 and a length of 60.

A possible pad stack setup is as follows:

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| ALL | | T | 20 | RF | 0 | 0 | 90 | 60 |
| | | I | 0 | R | | | | |
| | | B | 0 | R | | | | |
| 1 | 2 | T | 20 | RF | 90 | 0 | 90 | 60 |
| | | I | 0 | R | | | | |
| | | B | 0 | R | | | | |
| 8 | 11 | T | 20 | RF | 90 | 0 | 90 | 60 |
| | | I | 0 | R | | | | |
| | | B | 0 | R | | | | |
| 17 | 18 | T | 20 | RF | 90 | 0 | 90 | 60 |
| | | I | 0 | R | | | | |
| | | B | 0 | R | | | | |

NOTE: The INTDIA Field should be empty, no value entered.

How about an Edge Finger Connector? Say a 100 pin Edge Finger Connector with Odd Number fingers on the top layer and Even number fingers on the bottom layer.

You can do ALL the Fingers at the correct size and then individually enter the 50 that would be different from the ALL condition, but that is much effort. There is a better way. Simply, instead of using ALL, use ODD and EVEN. The following is a typical entry for an Edge Finger Connector.

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| ODD | T | 70 | OF | 0 | 90 | 400 | 0 |
| | B | 0 | R | | | | |
| EVEN | T | 0 | R | | | | |
| | B | 70 | OF | 0 | 90 | 400 | 0 |

Suppose you wished to create your parts as MULTILAYER parts in order that they could be used either for two layers or multi-layer applications?

In this situation, always have the database set to MULTILAYER, using the SETUP and MAXLAYER before starting to create your Part Decal. When you create the Decal, the PAD STACKS menu will be autoamtically set as follows:

Top Layer etc.

Inner Layers etc.
Bottom Layers etc.

The size and shape entered for the Inner layers are used to flash the oversize pads on Power and Ground Planes that are NOT tied into the plane. Power and Ground pins are tied into the planes using Thermal Relief pads that are identified in the Power (or Ground) net used in the photoplotting of the Power (or Ground) plane. (See Chapter on Multilayer Design for details.) However, the SIZE defined for inner pads is the SIZE used for the Thermal Reliefs.

Suppose your Multilayer board will be 6 layers with Layer 3 and 4 as planes, and 1, 2, 5 and 6 as routing layers using the regular size pads. In this instance, you would set the database to 6 layers, using SETUP, PARAMETERS and in PAD STACKS use the ADD LEVEL feature to add levels 2 and 5 to the Decal-Pad Stacks, and set Levels 1, 2, 5 and 6 at the same size/stack and I (Layers 3 and 4) as appropriate for Planes.

This concludes Tutorial 7. You should have a good appreciation of how to create a board database using either Parts Lists and Connection Lists, how to Design-On-The-Fly and finally, to create Part Types and Part Decals.

## Chapter 8

# TUTORIAL 7 - MULTI-LAYER DESIGN PROCEDURES

(180 minutes)
The first Tutorials were concerned with two-layer design procedures. In this Tutorial, we extend the training Tutorials to cover multi-layer design procedures by leading you through a four-layer design.

We will use the same database used for the two-layer design Tutorials, and in the process, we emphasize the differences between multi-layer and two-layer design.

## 8.1 Multi-Layer Setup (4 Layers) (Thru Via)

There is no difference in the initial data preparation phase between two-layer and multi-layer design. The Parts List is the same, the Connection list is the same, the reading of the two data lists into memory is the same and placement of parts is the same. Accordingly, call back your File XXX7 you created in Chapter 3, after you had read in the Parts and Connection List and placed the parts.

Select the SETUP (F2) routine. Select PARAMS (F4). The first step is to convert the two-layer database of XXX7 to a four-layer database. Select MAX LAYER and respond to the queries so as to create a four-layer board, with Layer 1 remaining as Layer 1 and old Layer 2 becoming new Layer 4. (Layer 2 and 3 will be used as Ground (Layer 2) and Power (Layer 3) planes.

Next, select DISPLAY and set the various items in DISPLAY to colors as appropriate for a four-layer board.

Next, all component Part Decal pads will require their pad assignments or criteria be set for multi-layer. Select PADS, and make a listing of

ALL Part Decal pad assignments on the board. Examine the listing and change Part Decal pad assignments as appropriate.
HINT: Pay attention to the Edge Finger Connector (P1). Fingers are required on Layers 1 and 4 but NO Fingers or Pads on Layers 2 and 3.

You do not need to concern yourself about Power and Ground Thermal pads on PWR and GND Planes of Layers 2 and 3. They will be taken care of in the post processing of the Ground and Power Planes.

Briefly, the following describes the creation of Power and Ground planes on multi-layer boards.

1. During design, do not route the Power and Ground connections, except for a specific reason such as to create a track leading from the finger of an Edge Connector to within the board area.

2. During CAM (Computer-Aided-Manufacturing) operations, select either Power Plane or Ground Plane when crating the Plane artwork (see the preceding chapter).

3. When the CAM menu requests:

**ENTER POWER/GROUND NAMES**

you enter GND for the Ground plane and +5V for the Power plane.

4. The system will then, in the photoplotter file, flash Thermal Relief pads on all Unrouted Ground Net pads on the Ground Plane layer and flash oversize pads for Routed Ground and Non-Ground pads.

You do need to insure that the NON-PWR/GND pins of Layer 2 and Layer 3 are set for planes, i.e., oversize pads, perhaps 70 mil Annular. After changing all the affected Part Decal Pads, make another LIST ALL, and double check for accuracy. The listing below is a partial listing of Pad Assignments on most of the Parts after the four layer assignment has been set.

PAD STACK LISTING

Finger Format -- Size Shape Orientation Length Offset Drill
Pad Format ----- Size Shape Drill
Shape = [OF]OvalFinger, [RF]RectFinger, [R]Round,
[S]Square,[A]Annular,[O]Odd

Pad Stacks for Part Decal: STANDARDVIA
Used by Part Types:

PIN 0 ( All pins not otherwise described )
TOP(1) 55 R 37
INNER 70 A 20 ( inner layers not otherwise described )
BOTTOM(4) 55 R 0

Pad Stacks for Part Decal: BURIEDVIA
Used by Part Types:

PIN 0 ( All pins not otherwise described )
TOP(1) 55 R 37
INNER 70 A 20 ( inner layers not otherwise described )
BOTTOM(4) 55 R 0

Pad Stacks for Part Decal: DIP14
Used by Part Types: 7404 7402 7474 7432
7440 7420 7400

PIN 0 ( All pins not otherwise described )
TOP(1) 60 R 37
INNER 70 R ( inner layers not otherwise described )
BOTTOM(4) 60 R

Pad Stacks for Part Decal: CON\26P\ED
Used by Part Types: CON\26P\ED

PIN 0 ( All pins not otherwise described )
TOP(1) 70 OF 90 400 0 0
BOTTOM(4) 0 R
PIN 2
TOP(1) 0 R 0
BOTTOM(4) 70 OF 90 400 0 0
PIN 4

```
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 6
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 8
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 10
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 12
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 14
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 16
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 18
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 20
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 22
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 24
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0
PIN 26
TOP(1)          0   R    0
BOTTOM(4)       70  OF   90    400  0


Pad Stacks for Part Decal:  XTAL1
Used by Part Types:  XTAL1

PIN 0  ( All pins not otherwise described )
TOP(1)          60  R    37
```

```
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
```

Pad Stacks for Part Decal: R1/8W
Used by Part Types: R1/8W,1K

```
PIN 0  ( All pins not otherwise described )
TOP(1)         60  R   35
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
```

Pad Stacks for Part Decal: DIP16
Used by Part Types: 74139

```
PIN 0  (  All pins not otherwise described  )
TOP(1)         60  R   37
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
PIN 1
TOP(1)         60  S   37
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
```

Pad Stacks for Part Decal: DIP20
Used by Part Types: 6167

```
PIN 0  ( All pins not otherwise described )
TOP(1)         60  R   37
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
PIN 1
TOP(1)         60  S   37
INNER          80  R   ( inner layers not otherwise described )
BOTTOM(4)      60  R
```

Pad Stacks for Part Decal: CL25
Used by Part Types: CL25,10UF

PIN 0 ( All pins not otherwise described )
TOP(1) 60 R 37
INNER 80 R 20 ( inner layers not otherwise described )
BOTTOM(4) 60 R 0

Pad Stacks for Part Decal: DCAP1
Used by Part Types: DCAP1,01UF

PIN 0 ( All pins not otherwise described )
TOP(1) 60 R 37
INNER 80 R ( inner layers not otherwise described )
BOTTOM(4) 60 R

Pad Stacks for Part Decal: MOIRE
Used by Part Types: MOIRE

PIN 0 ( All pins not otherwise described )
TOP(1) 200 O 0
INNER 200 O ( inner layers not otherwise described )
BOTTOM(4) 200 O
28 70 R

## PARTS PLACEMENT

The placement of Parts on the four-layer, multi-layer board is identical to the two-layer board.

## CONNECTIONS ROUTING

There are considerable similarities as well as differences between the two-layer and the multi-layer routing of the connections on the board. The Memory connections and Signal connections are converted to Horizontal Memory on Layer 1 and Vertical tracks on Layer 4 of the four layer board in the same fashion as Layer 1 and 2 of the two-layer board. The primary difference is the absence of Power and Ground

tracks on Layer 1 and 4 provides more routing area for the routing of Memory and Signal connections.

Perform the Placement/Routing Grid In Sync Test described in Section 4.5.1, and take action as necessary.

The only Power and Ground connections you need to route are from the Connector P1, pin 1 and 26, to the first part node to which these connections are connected. Route these Power and Ground connections on Layers 1 or 4 as appropriate. The remainder of the Power and Ground Nets are the Power and Ground planes and are generated automatically using the Ground/Power Plane feature of CAM.

When routing a four-layer board, the matched pair of Layer 1 and 2 for a two-layer board must be changed to a matched pair of 1 and 4 for the four-layer board. This means that when a feed thru or via is put in, the layer is automatically switched to Layer 1 or Layer 4, whichever may be the case. To change the Matched Pair layer, have the Information Pop-up Window displayed and use the Modeless command of:

PL 1 4 <CR>

and observe that the pop-up window message changes from PL 1 2 to PL 1 4.

## CHECKING

Air Gap Checking for a multi-layer board is identical to a two-layer board. Insure that all items "Errors" are visible on all layers.

## ALTERNATE METHOD

In the procedure for designing a Four Layer board described above, the suggestion was made that as soon as the database was created by reading in the Parts and Conn Lists, then using MAX LAYER, convert to a four layer board. Following this you were instructed to set the pad assignment for a four layer board.

The above procedure is the "approved" procedure. An alternative that may assist you especially during routing, is to NOT convert immediately

to a four layer board. Rather, leave as a two layer board until the Signal Connections are all routed and checked. At this time convert to a four layer board, then route ONLY the Power and Ground connections to the connector that must be on either Layer One or Four and leave the remainder of PWR and GND unrouted.

CAM

The post processing of artwork for Layers 1 and 4 are identical to Layers 1 and 2 of the two-layer board. Select Artwork, then Layer 1 or Layer 4.

Ground plane (Layer 2) and the Power plane (Layer 3) are somewhat different. For these items, be in CAM and in response to:

**SELECT PLOT TYPE**

select Power/Ground Plane. For Layer, select 25 and 2 (for Ground Plane). On the screen, in the sequence that reads:

**ENTER POWER (GND SIGNAL NAME)**

Enter GND. This creates the Ground Plane on Layer 2. Thermal Relief pads will be flashed on the GND Plane at all Unrouted Ground pins or Ground Pins that have both a routed track and an unrouted connection. The non-ground pins will be flashed on the plane as set in the Pad Assignment (80 R) for the Part Decal for Inner Layers.

NOTE: Thermal reliefs will be flashed on only Ground pins which have connections ONLY or a Track AND a Connection. If a Ground pin has a routed track only, a thermal relief will NOT be flashed. The significance of this is that normally Power and Ground planes should not have the connections routed, but rather should be left as unrouted connections.

Repeat for Layer 3 and VCC (+5V) to obtain the Power Plane.

Study your Matrix Printer plots for results.

## 8.2 Multi-Layers (In Excess of Four Layer) (Thru Vias)

When designing boards of 6-8 layers or more, consider the following:

(1) Change to Max Layers = 6. Old Layer 1 = New Layer 1; Old Layer Button = New Layer 6.

(2) Set up Pad Stack Assignment on all parts/vias for six layers: 1, 2, 5 and 6 as Routing Layers; 3 and 4 as planes (inner layers). Make a printout of pad stacks and compare to the following:

PAD STACK LISTING

Finger Format --- Size Shape Orientation Length Offset Drill
Pad Format ------ Size Shape Drill
Shape = [OF] OvalFinger, [RF] RectFinger, [R]Round, [S] Square, [A] Annular, [O] Odd

Pad Stacks for Part Decal: STANDARDVIA
Used by Part Types:

PIN 0 ( All pins not otherwise described )

| | | | | |
|---|---|---|---|---|
| TOP(1) | 55 | R | 37 | |
| INNER | 80 | R | 20 | ( inner layers not otherwise described ) |
| BOTTOM(6) | 55 | R | 0 | |
| 2 | 55 | R | 0 | |
| 5 | 55 | R | 0 | |

Pad Stacks for Part Decal: BURIEDVIA
Used by Part Types:

PIN 0 ( All pins not otherwise described )

| | | | | |
|---|---|---|---|---|
| TOP(1) | 55 | R | 37 | |
| INNER | 55 | R | 20 | ( inner layers not otherwise described ) |
| BOTTOM(6) | 55 | R | 0 | |
| 2 | 55 | R | 0 | |
| 5 | 55 | R | 0 | |

Pad Stacks for Part Decal: DIP14
Used by Part Types: 7404 7402 7474 7432
7440 7420 7400

PIN O ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM (6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

PIN 1

| | | | |
|---|---|---|---|
| TOP(1) | 60 | S | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM (6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

Pad Stacks for Part Decal: XTAL1
Used by Part Types: XTAL1

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

Pad Stacks for Part Decal: R1/8W
Used by Part Types: R1/8W,1K

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 35 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

Pad Stacks for Part Decal: DIP16
Used by Part Types: 74139

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

PIN 1

| | | | |
|---|---|---|---|
| TOP(1) | 60 | S | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

Pad Stacks for Part Decal: DIP20
Used by Part Types: 6167

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

PIN 1

| | | | |
|---|---|---|---|
| TOP(1) | 60 | S | 37 |
| INNER | 80 | R | ( inner layers not otherwise described ) |
| BOTTOM(6) | 60 | R | |
| 2 | 60 | R | |
| 5 | 60 | R | |

Pad Stacks for Part Decal: CL25
Used by Part Types: CL25,10UF

PIN 0 ( All pins not otherwise described )

| | | | |
|---|---|---|---|
| TOP(1) | 60 | R | 37 |
| INNER | 80 | R | 20 ( inner layers not otherwise described ) |
| BOTTOM(4) | 60 | R | 0 |

(3) Part Placement - same as 2 layer board

(4) Routing - Suggestion: Leave as a 2 layer board and route Layers 1 and 2. Then change to a 6 layer board, with old Layer One remaining as new Layer One, and old Layer 2 (assuming you start with a two-layer board) becoming new Layer Six. Note that Standard or Thru vias placed during routing now are Thru vias on all six layers. Use either Manual Routing or Automatic Routing.

(5) Select Layered Pairs 2 and 5, route Layers 2 and 5 and finish all Signal Connections

(6) Layers 3 and 4 are to be used as as Ground and Power planes.

(7) Run CHECK and correct any errors. Then plot all layers on the matrix plotter and check against your design at the CRT.

This exercise should leave you well acquainted with multi-layer design, using Power and Ground planes and Thru Vias.

## 8.3 Multi-Layers (Buried/Blind Vias)

Occasionally, you may want to design a multi-layer board using Blind or Buried Vias rather than Thru Hole Vias.

Blind Via: A via originating on an Exterior Layer and ending on an internal layer as established by the "Paired Layer Setting."

Buried Via: A via originating on an internal layer and terminating on another internal layer (established by the "Paired Layer Setting").

In this exercise you will design a six-layer multi-layer board using Blind vias.

Layer 1, 2, 5 and 6 are Routing Layers.
Layer 3 = GND Plane
Layer 4 = +5V Plane
Blind Vias are permitted between layers 1 and 2 and between layers 4 and 5.

Procedure:

1. Recall the file of the placed board ready to route, used during the early lesson (XXX7).

2. Set Pad Assignments for a six-layer board (i.e, ALL Routing levels have pad assignments of 60 mils). Use a Buried Via size of 28 mils Round, 15 mil hole.

3. Route the PWR/GND Connection from the connector (P1) to the first pin. Then turn OFF +5V and GND in NET ATTRIBUTE.

4. Set paired layers and Via Display in Information Window to read:
   PL1 2 V B
for layers 1 and 2 with Blind Vias between Layers 1 and 2.

5. Use ROUTE, AUTO RUT, Memory, Horiz/Vert, Short/Long Channel, one to eight vias, and Autorouter Layers 1 and 2.

NOTE: After routing on layers 1 and 2 with Blind Vias, inspect the results. Do you find vias on layers 3, 4, 5 and 6? You should not!

6. Set Paired Layers 5 and 6

7. Use ROUTE, AUTO and 1 Via to 8 Vias, Autoroute Layers 5 and 6.

8. In the event all the Signal connections are not autorouted on paired layers 1 and 2, or 5 and 6, use the SHOVE Router to route the remaining.

Note that in the event any of the unrouted connections are from the connector, you must route them on the layered pair corresponding to the layers on which the Edge Fingers of the connection correspond, i.e., Edge Fingers on Layer 1 connections must be routed on Layers 1 and 2 and Edge Fingers on Layer 6 connections must be routed on Layers 5 and 6. The fact that Blind Vias are used between Layers 1 and 2 and 5 and 6, but NOT between these pairs, dictates this.

The reason for this, of course, is the SHOVE Router is confined only to the layered pair indicated, and cannot shove tracks on the other layered pair to make room for the current layered pair.

In the event that the SHOVE Router does not complete the remaining connection, finish them interactively using ROUTE and RUT CON (F1), or SHOVE, AUTO PATH.

## 8.4 Tracks on Planes

The purpose of this section is to explain the process of routing tracks on planes. This will also explain the process of photoplotting these types of planes.

Occasionally, one has the choice of increasing the number of routing layers or else placing excess tracks on a PWR/GND plane. The reason for this is the very densely populated boards. Some companies do not want to increase the number of routing layers because of the increase in cost. In order to reduce the cost, it may become necessary to place some of the tracks on a power or ground plane, thus reducing the number of layers. The Electrical Engineer is usually the person responsible for making the decision as to which tracks can be placed on the inner planes.

The concept of including tracks on a plane is called "imbedded" planes. Prior to PADS-2000, imbedded planes was especially cumbersome and slow with potential errors. The concept required two different photo-plots--the first plot with tracks at their assigned width, the second with the tracks at some oversize width. After making the two plots as negatives from the photoplotter, one was made into a positive, then the two plots were superimposed on to each other and reprinted in order to make a composite piece of artwork, from which the imbedded planes could be manufactured. This procedure was expensive, time consuming and error prone.

The procedure you will now learn is unique to PADS-2000 and is possible only with the feature of COPPER POUR.

Let us proceed as follows with a new, easy and error free method of creating tracks on planes.

1. Bring up XXX7, the training board with components placed, ready to route.

2. Set the Board for 6 layers.

3. Set the Pad Stack Assignment for 6 layers
   1, 2, 5, 6 to be routed layers
   3 and 4 to be planes (3 = GND, 4 = +5V)

4. Turn +5V and GND Off, ALL On.

5. Now route a few of the Signal tracks on layer 3 to represent the tracks that you want to imbed into the Ground plane.

6. Select the CREATE Routine, the COPPER POUR routine. Using NEW POLY (F1), create a copper area over the entire board inside the Board Outline. Follow, using corners, the curved segment in the upper left corner of the board. Use COMPLETE (F9) to complete the COPPER POUR polygon. If this is a Ground plane, within the COP POUR routine select MODIFY, NETNAME, and select the EDGE of the Copper. At the prompt:

**Netname >**

enter: **GND**

Next select FLOOD (F8) to "flood" the area with copper.

Remember the flooding will stand off from pads and tracks by the amount of copper clearance. Suggest 20 mils.

7. Now check your efforts in SETUP. Make everything invisible on all layers except Layer 3 (the plane). You should see Layer 3 pads together with the tracks you wish to imbed in Layer 3.

8. Complete CAM, selecting Matrix Printer, Power/Ground Plane, Layer 3. On the menu indicating the items to be on the artwork, be sure to add tracks to the plots on Layer 3. Plot at 2:1 (for better inspection), enter GND, when requested.

The resulting plot will be a composite of plane and tracks, something that prior to COPPER POUR, would have required two different plots of the artwork: plane artwork and track artwork, then subsequent photographic merging, etc.

Repeat for Power (+5V) on Layer 4.

## Chapter 9

# TUTORIAL 8 - SMD DESIGN

## 9.1 INTRODUCTION

This tutorial will introduce you to the principles involved in designing when "Surface Mounted Devices" are used on the board.

The boards that will be designed will include all the requirements for SMD design. Two SMD Examples are included:

Example One
4 layers
Layers 1 and 4 for Signal routing and Layers 2 and 3 for Power and Ground Planes
Thruhole Vias
Parts on Top and Bottom Layers

Example Two
6 layers
Layers 1, 2, 5, 6 for Signal Connections
Layers 3 and 4 for Ground/Power planes
Blind Vias between paired layers 1 and 2 and paired layers 5 and 6 for Signal Tracks, Thru Vias used for tracks originating on Layer 1 and terminating on Layer 6
Thruholes for Power and Ground Planes

## 9.2 EXAMPLE ONE - 4-Layer, Multi-Layer

Call up the Board Outline we have prepared for you called SMDBO.JOB and display at the CRT.

The Parts List that we shall use is called SMDPARTS.ASC. To view, use ALT-9 and filename SMDPARTS.ASC or use the PRINT command

and print out. Note that the Parts List is similar to the Parts List used previously except the Part Types (for the most part) are SMD parts.

Into file SMDBO.JOB, read the Parts file SMDPARTS.ASC, using ASCII IN. Following this, read in the connections file called SMDCONN.ASC. Make a Job Save (S1).

## Placement of Parts

We will place the IC parts first. The IC parts need to be rotated until they are all Vertical with Pin 1 in the lower right corner. Do it now, Interactively.

We shall place the Memory IC's (U12-U21) on two matrices, one on the Top layer and the other on the Bottom layer in order to give you experience in designing boards with parts on both top and bottom layers (U12-U16 to be on the top and U17-U21 on the bottom).

### Bottom Layer

Using PLACE, AUTO, and MATRIX, Delete the existing matrix and make a new Placement Matrix with 1 row, Space of 200; 5 columns, Space of 700. It will be attached to the cursor. Move it to the bottom layer, using OPPOSITE, release it at approximately 2900 2200. You will need the Location Window turned On. This creates a matrix of five points located on Layer 2. (It appears as a straight line).

### Top Layer

Next, make an identical 5-point matrix except keep on Layer 1, and locate at about 500 1650. You now have a 5-point matrix on Layer 1 and a 5-point matrix on Layer 2.

## Auto Place of IC's

Next, place the Memory IC's using AUTO PLACE. Glue ALL down, then Interactively UNGLUE U12-U16 (the Memory IC's). Next, using OPPOSITE, move U17-U21 to the opposite layer.

The conditions of parts at this point should be ALL parts glued down except U12-U21. U12-U16 should be on Layer 1 and U17-U21 should be on Layer 2. Inspect for these conditions and make a Job Save. Let's now Auto Place IC U12-U21.

Using PLACE, AUTO and INITIAL PLACE, auto place U12-U21.

If all was done correctly, U12-U16 should now be located on the matrix on the top layer and U17-U21 should be on the matrix on the bottom layer. GLUE them down.

## Interactive Place of IC's

Place U1-U10 using Interactive Place. Unglue the IC's (U1-U10), and using SETUP, NET ATTRIBUTE, make the Connections VISIBLE (Display ON) and GND and +5V OFF. Set Grid = 50. Using MOVE, ROTATE and LENGTH MIN, locate U1-U10 according to their connection patterns. Sort out the IC's that have the largest number of connections originating at the connector and locate these IC's next to the connector immediately and above the connector pins to which they are attached. Continue until all IC's are placed. This is a good practical exercise in Interactive sorting out the location for a part by using its connections.

HINT: As you are trying a new location for a part, use LEN MIN (F3).

Next, place C1, C2, Y1, R1 and R2.

Use the Placement Evaluators to evaluate the ability of the router to put all the connections within the horizontal space between the two rows of non-memory IC's. If there is any question in your mind, spread the rows of non-memory IC's apart to take advantage of the space on the board.

Compare your placement to that of Figure 8.1a and b. Don't copy Figure 8 in your placement--make your own based on your judgement and we shall see how it routes. Make a Job Save.

## Setup

You want this to be a 4 layer board, therefore, using SETUP, PARAMETERS, make into a four layer board with Old layer 2 as New layer 4.

Next, using SETUP, DISPLAY, set the colors for all four layers as appropriate. Set the color of the connections to a different color from the layers.

Using SETUP and PADS, obtain a printout of all Pad Stacks. We see that the SMD IC's have a Pad Width of 25 mils. Since the Pads have a center line spacing of 50 mils, this leaves 25 mils space between the pads for track width and associated clearance.

What combination of track and space should we use? Use the formula:

$$T(\text{Width}) + (2 \times \text{Space}) \leq 25 \text{ between the pads}$$

A track width of 15 mils and 5 mil space would be satisfactory if one wanted to maximize track width:

$$15 + (2 \times 5) = 25$$

A track width of 8 and a space of 8 would be satisfactory if one wanted to equalize track and space.

$$8 + (2 \times 8) = 24$$

Use 8 and 8 in this practice.

In SETUP, SIZES set the 13 mil track width to 8. With a pad Width of 25 mils, you will want any Power and Ground tracks to be 25 mils. Therefore, change the 50 mil track to 25 mils.

In SETUP, PARAMETERS, set the Pad/Pad, Pad/Track and Track/Track clearance to 8, and the Real Width to 8. The latter will result in tracks being displayed at their "real" width.

Pad Stacks

Using SETUP, PADS, what is the size of the Thru Via? The Thru Via Decal is called STANDARDVIA. We see it is 55 mils Round, Top and Bottom Layers, with 37 mil Drill. This is rather large for fine line (8 mil track) design. Change to 30 mil Round, Top and Bottom with 20 mil Drill, and 60 mil Inner Layers.

We have completed SETUP. Make a JOB SAVE. We shall use this placement for both the 4-Layer example as well as the 6-layer example, so be careful to SAVE.

## Power and Ground Planes

Since for entry into the Ground and Power Plane, there will be short track segments from the SMD Parts on the top and bottom routing layers to Fan Out Vias, it is advisable to complete this phase of the board design before routing the signal connections. We shall do this now.

We shall be using Layer 2 as the Ground Plane and Layer 3 as the +5V plane.

A Review of Creating Planes Procedure

At this time, let's review the creation of Power and Ground planes.

Briefly, in PADS-2000, Planes are created as follows:

1. In the CAM Routine and plotting the artwork, on the Select Plot Type CRT screen, select Power/Ground Plane as the Plot Type.

Then select the Layer (level) for which the plot is to be created.

2. On the CAM screen that invites you to enter the Signal names of the nets that are to be included in the selected plane, enter GND. This selects the Net whose Nodes will be tied into plane selected above.

The results of (1) and (2) above accomplish the following:

1. All pins contained in the Net entered in (2) above are identified. Then action is taken on the layer (plane) designated in (1) above, as follows:

a) All pins in the Net that have one or more UNROUTED connections will have a Pad Stack Thermal Pad located on the plane, thus the pin will be tied into the plane.

b) All pins in the Net that have a mixture of Unrouted connections and routed tracks associated with the pin will also have a Thermal pad located on the plane, thus the pin is tied into the plane.

c) All pins in the Net that have routed tracks only, connected to the pin will have a pad flashed on the pin that is oversized at either the pad stack size set for inner layers or set specifically for the layer involved. In either case, the oversize pad, when plotted and converted to positive artwork for the plane will result in the pin NOT being connected to the plane.

d) All pads for all parts on the board including Vias, that are NOT in the selected net (GND) for the plane will be plotted with an oversize pad as determined by the inner layer size of the pad stack and when the artwork is reversed for the plane, will result in the pins not being connected to the plane.

With this background on the theory of creating Power and Ground planes, what is the procedure by which we create the plane?

First, recognize that if using Power and Ground planes rather than routed layers for the Power and Ground, then one does not, repeat not, completely route the Power and Ground connections, in the conventional sense of routing tracks. Instead, you proceed as follows:

1. Thru Hole Components

On all Thruhole components wherein you wish the pin to be connected through a thermal pad into the plane, simply insure that there is at least one unrouted connection to the pin, the software will automatically connect the Thermal pads when PWR/GND planes are created.

2. Edge Fingers

Edge Finger connectors will most likely have edge fingers on both the top and bottom layers, but no pads or edge fingers on the internal layers. For this reason you will have to provide some help. Here is how you proceed: Let's assume that Power is on Pin 1 of the Edge Finger. Make a short segment of track perhaps 0.2" long and then use the subroutine END (F8) to end the track as a Partial track with the standard thru via associated with the termination of a partial track. This partial track via is used during the plotting the the Power plane to connect pin 1 of the Edge Finger through the short Partial track you routed and then through the partial track via into the Power plane.

NOTE: For a pin or a via to be tied into a plane, the inner layers or the specific layer that is to be a plane must have its pad stack set such that there is a pad associated with that pad stack on the plane layers.

3. SMD Parts

The Power and Ground pads of an SMD are tied into a Power and Ground planes in exactly the same fashion as an Edge Finger. Specifically, you start routing the track away from the SMD pad, and when a short distance away from the pad, terminate the track with the subroutine END (F8), which results in a partial track via being placed at the end of the track. Be sure the length of the track between the SMD pad and the partial track via is enough to allow for a solder mask barrier to restrict the solder to the SMD pad contact area. Typically, this length should be in the order of 0.100" as a minimum.

4. Grounding of Unused or Spare Gates

In some designs, the designer may wish to ground the unused gates of an integrated circuit until the gate is used in some future Engineering Change Order. This can be easily done by routing a short track from the unused gate pin from either the top or bottom layer, out a short distance, and then using the END routine to terminate the track in a partial track via. This ties the unused gate pin into the ground plane until it is needed. When needed, simply make an ECO in the CAD board database and cut the track from the gate pin to the partial track via on the board.

## Creating the Power/Ground Planes

With the above background, let us proceed to create a Ground plane for our SMD Design. The Ground plane will be Layer 2. Using the routine SETUP and NET ATTR, turn ALL and +5V to Off and turn GND to ON. Upon exit, only the Ground net will be displayed. Run a Length Minimization to simplify the presentation of the ground net.

Using ROUTE, starting with U21 (upper left hand corner), select the ground connection (pin 10) of U21, and route out a short section (perhaps .2") on Layer 4 and terminate with the END routine. This will result in a short segment of track on Layer Four from pin 10 of U21 out to a Partial Track Via. Repeat for U18 through U21 (Layer 4).

NOTE: Memory connections, whether Horizontal or Vertical, have a regularly spaced pattern that may be interrupted with via pads. For this reason, when you are tying Power and Ground of the Memory IC's to plane, take particularly care to keep the Partial Track Via out of the pattern that will be used to route the memory tracks.

Repeat the same procedure for U1 through U16 (PIN 7) on Layer 1.

Note that you need take no action on thru hole components (Y1, R1, R2, C1, C2), assuming their pad stacks are set to create a pad on Layer 2; pads will be tied into the Ground plane when a Ground pin for these components exists in the GROUND Net; otherwise, these pads will be plotted oversize, and thus when "positive" artwork is made, the pins will NOT be tied to the Ground Plane.

Before proceeding further, you should check that all Ground pins have been tied into the Ground plane. This is performed in the CHECK routine and TIE PLANE (F5). Specifically, TIE PLANE (F5) identifies all the connections that have been identified as plane conditions (while setting up Net Attributes) that have NOT been tied into a plane. When TIE PLANE (F5) is selected, the prompt:

**Untied Plane Pin Support File Name:**

is presented. Respond with the name of the file that you wish the untied plane pins to be called. After the report is written, examine the file either by making a printout or using the ALT-9 command, then

indicating the name of the file. If all pins in the Ground net have been tied into the Ground plane, proceed on to the Power plane.

## Power Plane

After being satisfied with the Ground plane, proceed to create the Power (+5V) plane on Layer 3.

In NET ATTRIBUTE, make the +5V Net only Visible. Run a Length Minimization.

In ROUTE, select the Power connections and using END, terminate all SMD IC Parts' Power pins into a Partial Track Via with short tracks.

Tie the Power pin (P1) of the connector into a Partial track via with a short track on Layer 1.

Run CHECK, TIE PLANES and connect any unconnected power pins.

Once you are satisfied with the CHECK, TIE PLANE report results and have corrected any possible errors, use the CAM routine and plot both the Ground plane layer and the Power plane layer and inspect. We suggest plotting at a scale of 2:1 and including the component outlines and Reference Designations will make your analysis much easier.

## MEMORY CONNECTIONS ROUTING

Perform the Placement/Routing Grid Test described in Section 4.5.1, and take action as necessary.

You are now ready to route the memory connections on the board. Note that 5 of the Memory IC's are located on Layer 1 while the other 5 memory IC's are located on the bottom or Layer 4. You are to route the connections associated with the parts on layer 1 on Layer 1 and the connections associated with the parts on layer 4 on Layer 4.

You will recall that the earlier two-layer training board used the Memory Heuristic Router to route the memory connections. This is quite satisfactory for Dual in-line parts (DIPS) with pad stacks located 100 mils center-to-center. The Memory router, however, does not route

between pads that are located closer to each other than 100 mils. The Memory Heuristic Router is, therefore, not suitable for routing SMD board memory arrays.

A possible solution would be to use the Interactive router and interactively route the memory array. This is perfectly satisfactory. If you want a particular pattern for your memory connections you should route the memory array using Interactive Route.

We shall use the SHOVE Autorouter, in the AUTO PATH mode for routing the Memory array.

To AUTOROUTE memory connections, the standard strategy associated with the SHOVE Router must be altered. Let us do this now.

Select ROUTE, then SHOVE, then STRATEGY. While in STRATEGY, select MOD RULES (F5). At this point, take time out and as an introduction to what you will be doing, read the section on Strategy (Section 7.10, Chapter 7, Volume II of the PADS-2000 Reference Manual). After having read this introductory section, we shall alter the Strategy rules.

Select MOD RULES and study the CRT presentation and change STRATEGY as follows:

VIA COST: While routing Memory, we desire there be no vias. To do this, we make the via cost very high, therefore, change Via Cost to read 100.

RETRY COST: We do not want the Memory Router pushing and shoving other previously routed memory tracks to try to complete a connection. Therefore, set the Retry Cost relatively low. Set the Retry Cost to 15.

ROUTE LIMIT COST: We do not want the AUTO PATH Router to wander far afield or break other rules while routing simple memory connections. Therefore, we shall set the Route Limit cost relatively low. Set Route Limit cost to 15.

LEVEL DIRECTION FLAG: We wish to Enable routing on Layers 1 and 4 only, and Disable on Layers 2 and 3 (plane). Therefore, Enable Layers 1 and 4 and Disable Layers 2 and 3.

DIRECTION COST: Direction Cost on both Layers 1 and 4 should be 100.

Using AUTO PATH, start routing the top row of IC's, i.e., those that are on Layer 4.

You have profoundly changed the Routing Strategy with the above STRATEGY changes, and you should appreciate the significance. Basically, you have established a strategy that can be described as follows:

- Accept No Vias
- Connections on Pads on Layer 4 must be routed as tracks on Layer 4
- Connections or parts on Layer 1 must be routed on Layer 1
- You will not accept a complex track for a relatively simple, memory pattern.

Study "why" the changes you made in STRATEGY result in this effect. Proper use of STRATEGY can drastically affect the router results.

Here are some suggestions using the AUTO PATH Router while using Memory:

1. Start with the top pad on a row of IC pads and proceed down to the same row or start from the bottom and proceed up. Route all the connections on an IC before going to a second IC. Don't jump around.

2. Because of the type of "refresh" associated with a Personal Computer CRT, always refresh (Numeric key 1) after routing the connections associated with a row of pads. This insures that you can "see" all of the stubborn connection that otherwise might have been obscured by routed tracks.

NOTE: Occasionally, you may find, for some obscure reason, the AUTO PATH Router, while routing a straight forward simple memory connection tends to route in a "round about" fashion. Don't accept and don't spend time to second guess the system. Simply, interactively route the connections and proceed on.

After completing the Bottom row (Layer 4) connections, proceed to the Top layer and route these connections. Do not attempt to route the Vertical Memory I/O connections with the current strategy.

## ROUTING THE REMAINING SIGNAL CONNECTIONS

We will not be able to use the Maze Autorouter for the remaining connections inasmuch as they, too, are associated with components whose pads are spaced less than 100 mils. We will, therefore, use the SHOVE Router for the remainder of the connections. We must, however, change our Strategy from that used with the Memory connections, inasmuch as we will want to be routing on Layer 1 and 4 or the Top and Bottom layers, and we want the Autorouter to select the best possible path in a complex situation. Select STRATEGY and MOD RULES and change the strategy as follows:

VIA COST: In this instance, we will want to use Vias between the Top and Bottom layers, therefore give Via Cost an average cost. Set Via Cost to 50.

LEVEL DIRECTION FLAG: Ensure that Layers 2 and 3 are Disabled and Layers 1 and 4 are Enabled. Set Layer 1 Direction Cost to 75 (Horizontal) and Layer 4 Direction Cost to 25 (Vertical).

Starting from the connector, route the Signal connections using AUTO PATH. Observe the following while routing:

1. Try to make the routing window as large as possible, however, remember the larger the window, the longer (10-20 seconds) it will take to remap the routing window when moved.

2. Each time you Zoom, Pan or Exit the router, the routing window will be remapped.

3. A connection that is not entirely within the routing window will be routed within the window and not routed outside the routing window, and a Partial Track Via will be placed at the window limits. This Partial Track Via may have routing violations but they, of course, will be cleaned up when you route the remainder of the connections.

4. Some difficult connections may take as long as two or more minutes to "push" other tracks out of the way and find a suitable path. In the process, the word "Working" will be displayed. Remember, it would take a designer much longer to modify the existing tracks to make room for the track being routed and then route the track, than taken by the Shove Router. Despite this long time, you are ahead using the Autorouter, so let the AUTO PATH Router take its time to find a path.

5. While a connection is attempted to be routed, leave your cursor in the same position as it was at the time you selected the connection, such that in the event the Autorouter does not find a path the first time, you can select the connection a second (or even third) time. Except in "impossible" situations, this should result in routing the connections.

6. The Input/Output lines from the Memory array to the non-memory IC's can be difficult to route by the Autorouter. Here is a suggestion: Make sure the routing box includes both the Memory array as well as the Input/Output Lines to the non-memory components. In this fashion, when attempting to route the Input/Output lines, the Autorouter has the opportunity of running an internal Length Minimization and possibly find paths that otherwise would not be able to find, without this Global view.

NOTE: Unless you have a particular desire to do so, only route enough of the additional non-memory signal connections to assure yourself that you understand the principle and the techniques involved. Make a Job Save and proceed on.

CHECK

1. Space Check. Using SETUP, DISPLAY, set the colors of the errors on all four layers different from the tracks and pads on the associated layer. In this fashion, any errors will be more visible. Run a spacing check and correct any problems.

2. Connectivity Check. Using CHECKING and CHECK ASCII (F2), run a check against the input connection file (SMDCONN), and correct any problems.

3. Plane Checking. Using CHECKING and TIE PLANE (F5), run a check to ensure that both the +5V and the GND nets are completely tied into planes.

## CAM

Using the CAM Routine, plot all four layers on your matrix printer. Do so at a scale of 2:1 and also plotting the Component Outline and REF DES. Visually inspect all layers, relating them to each other to ensure that you understand the multi-layer procedure. The four layer plots provide an excellent visual aid for your appreciation of multi-layer design techniques.

## 9.3 Multi-Layer Board, Example #2

The remainder of this chapter will be concerned with the design of a six layer, multi-layer board.

In this example, Layers 1, 2, 5 and 6 will be used for signal connection routing, and Layers 3 and 4 will be used for Ground and Power planes. We will introduce the concept of Blind vias. Blind vias will be used between Layers 1 and 2 when routing connections on these two layers, as well as between Layers 5 and 6 when using these two layers as layered pairs; they do not project through all six layers.

In the process of designing the board, we shall use the same placement as was used for Example #1. If you save the "placed board" of Example #1 after the parts were placed, but before the Power and Ground planes were created and the signal connections routed, then call up this file and display it at this time. If you did not save this file, then read in the Board Outline (SMDBO.JOB), and into the Board Outline, read in the Parts List (SMDPARTS.ASC) and the Connection List (SMDCONN. ASC). Following this, place the parts in the same fashion as was used to place the parts for Example #1.

## SETUP

SETUP the placed board as a six-layer board. Using SETUP, PARAMS, set the REAL WIDTH, such that the routed tracks will be displayed at their Real Width. Set Pad-to-Pad, Pad-to-Track, and Track-to-Track clearances to 8 mils. Set the maximum routing level to 6. Set Old Layer 1 to New Layer 1 and Old Layer 2 to New Layer 6.

Using SETUP, DISPLAY, set all six layers to a unique, identifiable color. Set the Connections to a color that is uniquely different from the 6 layers (white, for example). Set the TOP item to the same color as Layer 1 and the BOTTOM entry to the same color as Layer 6. Set the Part Name, Layer 1 to the same color as Layer 1, and Part Name, Layer 6 to the same color as Layer 6. Exit and inspect the results at the CRT to ensure that you have set the display as intended.

Using SETUP, PADS, ADD LVL, set the Decal called STANDARD VIA as follows:

| | | | | |
|---|---|---|---|---|
| ALL | T | 30 | R | 15 |
| | 2 | 30 | R | |
| | 3 | 60 | R | |
| | 4 | 60 | R | |
| | 5 | 30 | R | |
| | 6 | 30 | R | |

(NOT in the above order)

Next, set the Decal called BURIEDVIA the same as for STANDARD VIA.

Finally, make a LISTALL Listing of all Decals on the board and except for the Edge Finger connector, set for six layers as you have done with STANDARDVIA and BURIEDVIA Decals. When complete, make another LISTALL and carefully study the printout to ensure you have the Decals set for a six-layer design. Make a JOB SAVE.

NOTE: Be aware that Partial Track Vias and Thruvias will penetrate all six layers of the board whereas Blind Vias will penetrate only the layered pairs at the time the Via was added. The significance of this is that during Board Manufacturing, Layers 1 and 2, using Blind Vias should be built and drilled as a layered pair. Similarly, Layers 5 and 6 should be built and drilled as a layered pair. Then all six layers will be assembled, and the Thruvias and Thru Holes will than be drilled. You will need to create a drill drawing/tape for paired layers 1 and 2, paired layers 5 and 6, and then for the entire board, and provide this to your Board Manufacturer with complete drilling information.

## POWER AND GROUND PLANES

We shall now create the Power and Ground planes. The Ground Plane will be Layer #3 and the Power Plane (+5V) will be Layer #4.

Using SETUP, NET ATTRIBUTE, set All to Display OFF. Then set +5V and GND to Display ON, Length Min Type to Vertical and Plane to Yes. Exit and using PLACE, AUTO, run a Length Minimization. Using the same rules as used for the first example, from both the SMD Power and Ground Pins and the Edge Finger Connector, route short lengths of track (.2" to .3" in length) and terminate with a Partial Track Via, using the END subroutine of the ROUTE Routine.

Recall that on Thru Hole components, such as the Crystal, Caps and Resistors, you do not need to take routing action for the Power and Ground Planes. Simply leave connections going to these pads as unrouted connections.

When both the Power and Ground Plane Partial Track Vias have been completed, use CHECK, TIE PLANE and run a check and inspect the resulting report to insure that all Power and Ground pads of all components on the board have been tied into the Power and Ground Planes. Following this, plot Ground Plane (layer 3) and +5V (layer 4) on the matrix printer, at a scale of 2:1 and inspect the results.

### Memory Array Connection Routing

When routing the Memory connections, you should proceed exactly as was done for the four-layer board, except that you are routing on Layer 1 and 6, rather than 1 and 4. Select ROUTE, SHOVE, STRATEGY, then MOD RULES. Set STRATEGY as follows:

VIA COST - Set to 100
RETRY COST - Set to 15
ROUTE LIMIT - Set to 15
LEVEL DIRECTION - Enable Layers 1 and 6
Disable Layers 2, 3, 4 and 5
DIRECTION COST - Layers 1 and 6 to 100

With the Strategy, you are in a position to route Memory connections on Layer 1 located parts on Layer 1 and Layer 6 located parts on Layer 6.

Proceed to route the Layer 6 Memory connections, followed by the Layer 1 Memory connections.

Non-Memory Connection Routing

We will now route the non-memory connections. In the process, we must (1) set up the layered pair and Blind Vias, and (2) set up the proper routing strategy.

Layered Pairs and Vias

Select ROUTE and then change the line in the Information pop-up window that currently reads PL 1 2 V T to read PL 1 2 V B. Use a Modeless command. Simply enter V B <CR> and inspect that the line reads PL 1 2 V B.

Then SHOVE, then STRATEGY, then MOD RULES.

Set Layers 1 and 2 as Enabled and Layers 3, 4, 5 and 6 as Non-Enabled.

Using AUTO PATH, and starting at the P1 connector, route as many of the connections as possible on Layers 1 and 2.

NOTE: Connections that originate on Layer 6 of the P1 Edge Finger connector or originate on Layer 1 of the Edge Connector and terminate on Layer 6 of an SMD component will not be selected for autorouting inasmuch as all routing of a connection must originate and complete on routing pair Layers 1 and 2.

After the connections on Layer 1 of the Edge Finger have been routed, then proceed to route as many of the remaining non-memory connections as desired. Those that originate and complete on Layers 1 and 2 will be routed. Connections that either originate or complete on Layer 6 will not be routed.

Paired Layers 5 and 6

When you have routed as many of the non-memory connections as possible on Layer 1 and 2, trying to balance the track between 1/2 and 5/6. Exit the SHOVE Router and while in the ROUTE routine, change

the routing pairs to Layers 5 and 6. The line in the Information window should read PL 5 6 V B.

Enter the SHOVE Router, STRATEGY, and MOD RULES, and under Level Direction Flag, Disable Layers 1, 2, 3 and 4 and Enable Layers 5 and 6.

Starting from the P1 connector, route as many of the remaining connections that terminate the Edge Connector on Layer 6 as possible, then proceed to route as many of the remaining connections on the board.

All Routing Layers

You will not have been able to route the connections that originate/complete on Layer 6 and originate/complete on Layer 1, using Blind Vias. Eventually, when you have a connection originating on the top layer and completing on the bottom layer, you must use Thru Vias, and pair up Layers 1 and 6. Do this now to finish all remaining connections.

Set Layered pairs to 1 and 6 (PL 1 6).

Set Vias to Thru Vias (V T)

In ROUTE, SHOVE, STRATEGY, Enable Layers 1, 2, 5 and 6. Disable Layers 3 and 4.

The above will permit the strategy of a track on Layer 1, routing a portion on Layer 2, routing a portion on Layer 5 and completing on Layer 6 (or any combination thereof) using Thru Vias.

Finish routing all connections on the board.

CHECKING

Check for SPACING, ASCII IN and TIE PLANES. Resolve any problems.

## CAM

Plot all six layers plus a Solder Resist Mask for both Top and Bottom Layers and analyze.

## CONCLUSION

This concludes your tutorial on SMD Design. You have been exposed to most of the situations that will occur in Multi-layer SMD Board Design. With this exposure, as additional SMD Design requirements surface, you should be able to utilize your PADS-2000 to the best advantage in meeting these new requirements.

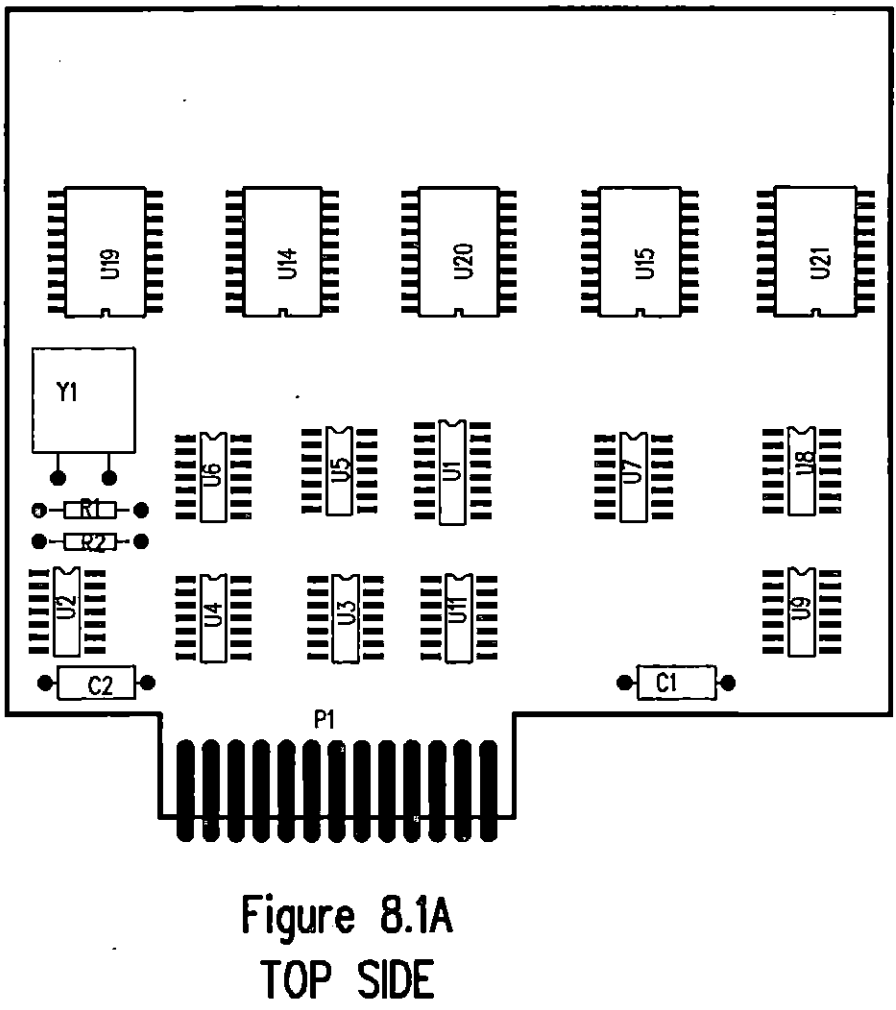



*Figure 8.1a - Top, SMD Training Board*

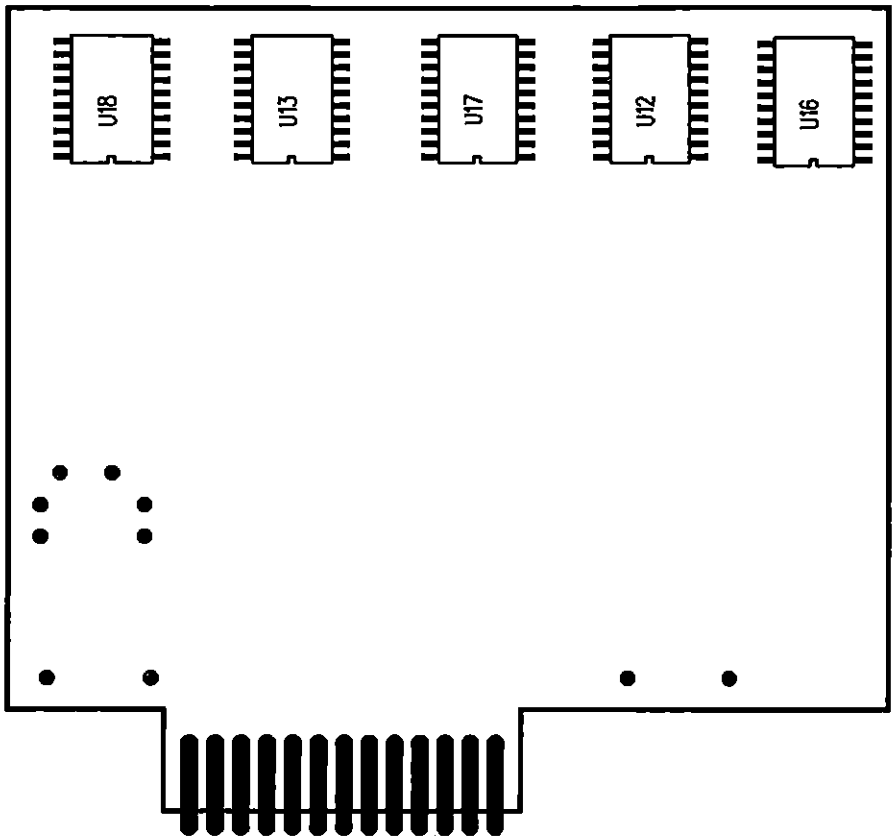



Figure 8.1B
BOTTOM SIDE

*Figure 8.1b - Bottom, SMD Training Board*

## Chapter 10

# TUTORIAL 9 - JOB FILES AND ARCHIVAL MAINTENANCE

There are two sets of files that you need to "guard like gold." The first are the .JOB and .ASC files representing the database of the board you have designed, or else are in the process of design. These files are maintained in the subdirectory called FILES.

The second set of files are the files that comprise the System Library, i.e., Part Types, Part Decals and 2-D Lines. These files are maintained in the subdirectory LIB.

DATABASE FILES
(See Volume II, Chapter 14)

As you will recall, during the series of training exercises, you made a number of database "saves" at different stages. So it is with the design of a board. The intermediate database "saves" are made to hard disk in either Binary format (.JOB) or ASCII (.ASC).

These files take up space on the hard disk and when the board is finished, need to be removed from the hard disk. Incidentally, for space estimating purposes, you can use one KB of memory or disk space for every " 14 pin equivalent IC" on the completed board design.

In order to decide on what to delete or clean off the hard disk, you need to know what is on it. Let's enter the directory of FILES and see.

Type:

**CD\PADS\FILES <CR>**

Response:

**C:\PADS\FILES>**

Type:

**DIR**

(for Directory Listing)

A listing of all the files in the subdirectory FILES is displayed including File Name, Type, Size and time of creation. Alternately, enter JOBIN and respond * <CR> and a pop-up window will list all files. Decide on the files that you want to keep and the files you want to delete. In general, you will want to keep the last file created during the design; that is, the file used to create the artwork and documentation. This file is called the ARCHIVAL FILE and is transferred to floppy disk for long time storage.

Let's practice these deletes and transfers.

FILES XXX1 through XXX12 (where XXX are your initials) represent interim files that can be deleted from the hard disk. File XXX12 represents your final design and will be treated as the ARCHIVAL FILE.

FILE DELETION

Be in the FILES directory, the directory that reads C:\PADS\FILES. Make a listing using DIR <CR> Write out those files you wish to delete, i.e., XXX1 through XXX11. To delete file XXX1, type:

**DEL XXX1.JOB <CR>**

The file XXX1.JOB will be deleted. Run DIR again to check the deletion.

In a non-training situation, you would, upon completion of a design, delete all the files associated with the design except the ARCHIVAL FILE.

ARCHIVAL FILE STORAGE

FILE XXX12, your last file during the basic training Tutorial, will be treated as the ARCHIVAL file and placed on floppy disk for long term storage.

Have a blank formatted high density floppy disk in DRIVE A, the floppy disk drive. Be in the FILES directory.

In response to:

**C:\PADS\FILES>**

Enter:

**COPY C: XXX12.JOB A: <CR>**

To transfer File XXX12 from Drive C to Drive A. After the transfer is complete, check it by making Drive A the Default drive and use DIR to obtain the directory of the contents of the floppy on Drive A. If the transfer was satisfactory, the floppy should contain the file XXX12.JOB.

At this point, you could delete XXX12 from the hard disk if you needed more disk space.

Caution: Job Files resulting from PADS-2000 are larger in size than those created in PADS-PCB. Moreover, you may be using PADS-2000 on larger boards than could be designed on PADS-PCB. As a result, you may find that even high density floppy disks will not be sufficient to hold a very large board of say 800 IC's and you will require two disks. In this instance, use the DOS command of BACKUP and RESTORE. An alternate solution is one of the public domain Packing Programs. CAD Software uses PKZIP and finds it satisfactory.

SYSTEM LIBRARY FILES (See Volume II, Chapter 12)

From time to time, as you add Parts or Part Decals to the System Library, you will want to make floppy disk backup copies of your Library. In fact, we recommend you make a backup every time you enter a significant number of new parts. To save your System Library, proceed as follows:

Have a fresh formatted floppy disk in Drive A.
Enter the subdirectory LIB with the command:

**CD\PADS\LIB**

The response is:

**C:\PADS\LIB>**

PART TYPE COPY

You enter:

**COPY UPART.DAT A:<CR>**
**COPY SPART.IDX A:<CR>**

This will copy four files from your hard disk to floppy Drive A: SPART.DAT, SPART.IDX, UPART.DAT and UPART.IDX. These are the Part Type file and associated Index file for Part Types.

PART DECAL COPY

To copy the existing Part Decal records FROM the subdirectory LIB TO a floppy disk, proceed as follows:

Place a floppy in the destination drive (in this example, drive A:), enter the subdirectory LIB by entering the command:

**CD\PADS\LIB <CR>**

Response:

**C:\PADS\LIB>**

You enter:

**COPY UPCBDECAL.DAT A:<CR>**
**COPY SPCBDECAL.IDX A: <CR>**

This will copy four files from your hard disk to floppy drive A: SPCBDECAL.DAT, SPCBDECAL.IDX, UPCBDECAL.DAT and UPCBDECAL.IDX.

**Caution**: If you backup the Decal Library, the Lines Library MUST be backed up as well. This is because there are "pointers" to and from the Decal and Lines Library.

LINE ITEMS TRANSFER

To copy the existing Line Item records FROM the subdirectory LIB TO a floppy disk, proceed as follows:

Place a floppy in the destination drive (in this example, drive A:), enter the subdirectory LIB by entering the command:

**CD\PADS\LIB <CR>**

Response:

**C:\PADS\LIB>**

You enter:

**COPY ULINE.DAT A:<CR>**
**COPY SLINE.IDX A:<CR>**

This will copy four LINE files from your hard disk to floppy drive A: SLINE.DAT, SLINE.IDX, ULINE.DAT and ULINE.IDX.

NOTE: 1 With a very large system library, you may want to keep Part Types on one floppy disk and Part Decals and 2-D Lines together on a second disk.

NOTE: 2 Always transfer together the Part Decals and 2-D Lines from the hard disk to the floppy disk, and then together back to the hard disk.

BACKUP FILES

We conclude our Self-Training on a happy subject: the BACKUP security files and its usage!

Every five minutes (unless changed in SetUp), a copy of the current database resident in memory, is written to a reserved space on hard disk, overwriting the previous contents. This is called a Backup or Security file.

Should for any reason your system go down or lock up as a result of power failure, hardware failure, software bug, cockpit error--any reason whatsoever--don't panic!! Simply reboot or turn the system OFF and then ON and reboot. When PADS-2000 is initialized, Select IN/OUT, then JOB IN. Then strike the ENTER key. The last Backup file made before the failure occurred will be brought into memory, ready for your use. And you will have lost only about five minutes of work.

# Chapter 11

# TUTORIAL 10 - THE MACRO ROUTINE

## 11.1 Introduction

In this chapter, you will become acquainted with Macros, how to create them and how to use them. Macros are a form of shorthand; that is to say, many individual commands can be combined into a single command, thus saving considerable effort on your part.

The PADS-2000 Reference Manual, Chapter 4, describes Macros in detail and can be referenced when needed.

The MACRO Management routine is located in the SETUP subroutine as MACROS (F6). When selected, the menu changes to the following:

**1 Status 2 Load/File 3 Save 4 Del MAC**
**10 Exit**

STATUS (F1), when selected, provides a Status file of the Macros using the ALT key.

LOAD FILE (F2) is used to bring into memory a file that you have previously created and saved, and intend to use again.

SAVE (F3) allows you to save a macro under a macro file name.

DEL MAC (F4) allows you to delete selected or all macros.

## 11.2 Macro Commands

There are some basic Macro commands that you MUST know. These are as follows:

ALT 1 = Replays Auto Macro
ALT 2 = Create/Finish Auto Macro
ALT 3 = Create/Finish User Macro
ALT 4 = Set Coordinate Mode (Absolute/Relative)
ALT 5 = Not Used
ALT 6 = Set Yes/No Mode
ALT 7 = Restore Menu Position
ALT 8 = Go to Top Menu
ALT 9 = Display File
ALT 0 = Display Errors (PCB.ERR)
C = Replay Count for User and Auto Macros

NOTE: ALT 2 and ALT 3 both start AND finish the recording of a Macro.

NOTE: These macro commands are listed under STATUS (F1), under MACROS.

## 11.3 Types of Macros

There are a number of types of Macros:

- Auto Macros
- User Macros
- Menu Macros
- Modeless Macros

The first three types are true macros and can be created by yourself using the MACRO feature. Modeless Macros are a special form of macros and are provided as part of the PADS-2000 software. They are created by CAD Software, Inc.; facilities are not provided in PADS-2000 to permit the user to create Modeless Macros.

## 11.4 Creating Macros

You can create a macro using two different techniques:

1. By performing the action at the CRT/keyboard and recording the macro.

2. By using a Text Editor and composing a Macro file in Macro language.

In this chapter, you will use both methods.

## 11.5 Creating an Auto Macro

What is an Auto Macro? An Auto Macro does not take a name and hence only one can be created at any given time. (The other type of macros, the ones with names, are called User Macros). The Auto Macro exists mainly as a quick way to make and execute a temporary macro; this macro cannot be saved to a file, and thus cannot be reused.

Another difference between the Auto and User macros is that at the start of Auto Macro creation, the Macro system is automatically placed in Relative Coordinate mode.

Auto Macros are created by selecting ALT 2, at which time macro recording starts. When the macro is completed, selecting ALT 2 once again, completes the macro and stops the recording. To replay the Auto Macro, select ALT 1.

NOTE: Only one Auto Macro may exist at a time. The selection of CREATE, MACRO (ALT 2) overwrites the existing Auto Macro.

Creating an Auto Macro

In this exercise we shall create an Auto Macro. This Auto Macro will be capable of adding any number of terminals (parts) while creating a Part Decal.

Here are the steps to accomplish this Auto Macro:

You may be anywhere within the PADS-2000 software.

1. Select ALT L (to enter the library routine).
2. Select CREATE (F1).
3. System prompts:
   **Part Decal Name <CR = Test>**
   Strike Return.
4. Move cursor to the Origin (0 0) by typing, S0 0 <CR>

5. Select Terminal (F3)
6. Select CTRL L to display the X and Y coordinates.

Now we start recording the Auto Macro.

7. Select ALT 2 (to start recording the macro).
8. Select ADD TERM (F1).
9. Select COMPLETE (F1).
10. Move the cursor to the X coordinate 100 and the Y coordinate 0. (Do not use the Search command.)
11. Select ALT 2 (to complete the recording of the Auto Macro).
12. To execute the Auto Macro, type C9 and select ALT 1. (This will allow the macro to be executed 9 times.)

There should now be a row of 10 terminals on the CRT. Let's now move the cursor to a new set of coordinates.

13. Move the X coordinate 0 and the Y coordinate 100.

14. Type C10 and select ALT 1. (This will now add a second row of 10 terminals).

As you can see, this macro can be a great time saver. Try your hand at making an Auto Macro for a 96 pin connector (2 rows of 48 pins each--.156" apart in the X direction and .100" between rows).

NOTE: While you are recording the Macro, your every action is also recorded. For this reason, we suggest that for the more complex Auto Macros, you write out an outline of the required steps you intend to do before starting the macro.

## 11.6 User Macros

User macros are those macros that you expect to repeatedly use over and over (as differentiated from an Auto Macro) and, therefore, after it is created, you save it for future use. The User Macro is always saved in a file called PCB.MAC.

While ALT 2 was the START/STOP command for Auto Macros, the Start/Stop command for User Macros is ALT 3.

The User Macros listed below are contained in a file called PCB.MAC. This file is located in the \PADS\FILES directory. If the macros are accidentally deleted, a backup file of PCB.MAC has been included. The file name is PCB.SAV and this file resides in the \PADS\FILES directory. Simply use the DOS copy command: COPY PCB.SAV PCB.MAC

Practice: Go to various positions of the menu routines and subroutines and then use the different Macro commands until you are sufficiently familiar with them, so as to be able to use them to do productive work.

NOTE: If the user has entered PADS-2000 by utilizing "the Shell," ALT X will exit the user to the Shell. If the user has entered PADS-2000 by typing "PADS2000," ALT X will exit the user to DOS.

## 11.7 Macro Language

Macros can be created as a text file using a Text Editor and Macro Language. The latter is quite obvious and can be illustrated with some examples.

The following are text files of the Menu Macros you used in the preceding section. They have been created using the PEDIT Text Editor.

```
(bdef) (*EXIT*) (a-x)      Begin definition, macro name, alt x
(yn)y                      Set Yes/No mode to Yes
(top)                      Top menu
(f10)                      Exit
(edef)                     End of definition

(bdef) (*DISPLAY*) (a-d)
(top)
(f2)                       Setup
(f1)                       Display
(edef)

(bdef) (*FLOOD*) (a-f)
(top)
(f3)                       Create
(f8)                       Flood
```

```
(edef)

(bdef) (*LIBRARY*) (a-l)
(top)
(f3)                    Create
(f5)                    Part Decal
(edef)

(bdef)  (*MACRO*) (a-m)
(top)
(f2)                    Setup
(f6)                    Macros
(edef)

(bdef) (*JOB IN*) (a-j)
(top)
(f1)                    In/Out
(edef)

(bdef) (*ECO*) (a-e)
(top)
(f7)                    ECO
(edef)
```

These Menu Macros are executable from any routine within PADS-2000. The User simply decides which command is needed, and types the ALT character, and the Menu Macro is executed.

Practice: Using PEDIT or other Text Editors, create a text file for a Menu Macro that will take you to the ROUTE subroutine from any location in the PADS-2000 menu structure.

## 11.8 Loading User Macros

Macro files can be loaded two ways. One way is to put the User macros that you always want loaded in the file, PCB.MAC. This file is located in the \PADS\FILES directory. This file is automatically loaded at program startup.

The second way to load a macro is to load from within PADS-2000 by using the LOAD command in the Macro Management menu. The LOAD

command will always load all the macros in a file. If any of the macros in the file already exist in memory, the loading process will be aborted.

## 11.9 User Design Macros

User Design Macros are those macros that are of a nature that their use is so frequent during a design that they are saved or stored and then recalled when needed. Generally a User Design Macro is associated with a Design feature such as creating parts or quickly entering another menu routine.

In the following exercise, you will create a User Macro that will bring you into the ROUTE routine from anywhere within PADS-2000.

1. Select ALT 3 to start recording the User Macro.
   Screen prompts:
   **What key will you use to invoke this macro?**
2. Select ALT R.
3. Next, give a description of the macro using 11 characters or less.
   Enter: **Route <CR>**

NOTE: Using a space counts as a character.

   Screen prompts:
   **Now creating macro (a-r)**
4. Select ALT 8 to get to the top menu. (ALT 8 is an existing macro.)
5. Select F5 for the ROUTE routine menu.
6. Select ALT 3 to stop recording the User Macro.
7. Screen prompts:
   **Macro (a-r) created.**

The macro must now be saved to a macro file.

1. To accomplish this, select ALT M.
2. Select SAVE (F3).
3. Output filename? (CR=) Type PCB1.MAC <CR>
4. Screen Prompts:
   **Macros saved to file c:\PADS\FILES\PCB1.MAC**
5. Now select ALT 8, and this will bring you to the top menu.

## 11.10 How to Replay a User Macro

To replay this User Macro for ROUTE, simply select ALT R from anywhere within PADS-2000 and the Menu will be brought to the ROUTE routine.

## 11.11 What are Menu Macros

MENU MACROS are a special kind of USER MACRO that allows the PADS-2000 User to select a menu command quickly with one key stroke. The User does not need to select menus, then submenus in order to find the executable menu item. By selecting the Menu Macro, the PADS-2000 User will be "jumped" to the desired menu item and ready for the next command.

Some menu command macros that have been already created are as follows:

ALT E = ECO Menu
ALT X = Exit to DOS or the 2000 Shell
ALT M = Macro Menu
ALT L = Library Menu
ALT F = Flood Copper Pour
ALT D = Display Menu
ALT J = Job in Menu

The various Menu Commands listed above are single key commands and allow the User to enter the ECO routine, Exit completely to the DOS prompt, get to the MACRO menu item, moves the User to the JOB IN menu, and to the Library Menu, to Flood Copper Pour and to the Display Menu.

# INDEX

## A

ABORT 59
ADD 52
ADD CONN 123
Add Connection 123, 124
Add Part 124
ALTERNATE 63
ALTERNATES 150
Aperture 135
Aperture assignments 136
Aperture Table 136
ARCHIVAL FILE STORAGE 206
ASCII files 19, 139
ASCII IN 19, 141
ASCII INPUT 139
ASCII OUT 6, 19, 142
ASSEMBLY DRAWING 134
Assignment logic 137
Auto Macros 212
AUTO PLACE 64
AUTO RENAME 79
AUTO RUT 91
AUTO STEER 102
AUTOPLACEMENT 74

## B

BACKUP INTERVAL 27
Blind vias 5
Board Edge Clearance 31
BOARD MECHANICAL 44
BOARD OUTLINE 49
BOTTOM 23
BOX WINDOW 33
Buried vias 5

## C

CAM 6, 131
CENTER 34
CHANGE ORDERS 117
CHECK 5
Checking 5, 117
CHKDSK/F 10
CLEARANCE 27
Command Menu 14
COMPONENT PLACEMENT 81
COMPUTER MANUFACTURING AIDS 131
Connection 5
CONNECTION LIST 141
CONNECTION ROUTING 89
CONNECTOR PLACEMENT 68
CONTRACT 34
CONV IN 19
Copper 5, 106
COPPER HATCH GRID 27
COPPER POUR 26, 110
COPPER POUR CLEARANCE 27
Crash 10
CREATE 5, 50
CREATE ROUTINE 49
CREATING 2-D LINES 52
CREATING A BOARD DATABASE 54
Creating an Auto Macro 213
Creating Macros 212
CRT PRESENTATION 14
CTR-L 18
CTRL-I 18
CTRL-L 14
CURRENT NET HIGHLIGHT 25
Cursor Position indicator 50

**D**

D code 136
Data Check 119
DATABASE FILES 205
Decoupling Cap 105
Default Track Width 31

Delete Part 123
DENSITY MAP 78
DESIGN CRITERIA 45
DESIGN-ON-THE-FLY 139
DESIGNING-ON-THE-FLY 144
DIRECT 132
DISCRETE PARTS PLACEMENT 75
DISPLAY 22
Display List 11
DOT GRID 26
DRILL DRAWING 134

**E**

ECL 29
ECO 5
ECO Report File 80
ECO's 120
Engineering Change Orders 5, 120
Erase Job 19
Error messages 14
ERRORS 118
EXPAND 34

**F**

From the Schematic 121
Function Keys 16
FutureNet 19

**G**

GATE AND PIN SWAP 76
Gerber 131
GLOBAL GLUE 64
GLUE 63
Ground 91
Ground plane 168
GROUP OPERATIONS 80

**H**

HEURISTIC ROUTER 108
HISTOGRAM 78
HORIZ/VERT 108
Horizontal 29

**I**

IC PLACEMENT 73
IN/OUT 4
Information window 18
INPUT/OUTPUT 19
INTERACTIVE PLACE 58

**J**

JOB FILES 205
Job In 19, 20
Job Out 19, 35
Job Save Log 36

**L**

Laser photoplotters 135
LEN MIN 104
LEN OF 92
LENGTH MIN ON THE FLY 62
Length Minimization 29, 107
Level Direction Flags 31
LIBRARY PARTS 146
LIMITS 117
"Location" window 18
LONG CHANNEL 108
Long Memory 108

**M**

Macro Commands 211
MACROS 30, 211
MAN PATH 103
MATRIX MOVE 60
Matrix printer 131
MAX ROUTING LEVEL 28

MAZE ROUTER 109
Menu Macros 212
Minimum 29
Minimum Track Clearance 31
MODELESS COMMANDS 35
Modeless Macros 212
MODIFY 92
MODIFY TRACK 97
MOIRE PAD 69
Mouse 11
MOVE 59
MOVE NAME 64
Multi-layer 167
MULTI-LAYER DESIGN PROCEDURES 167
Multi-Layer Setup 167
Multi-Layers (Buried/Blind Vias) 178
Multi-Layers (In Excess of Four Layer) 175

**N**

NET ATTR(IBUTE) 29
NET ATTRIBUTES 30, 104
NEW POLY 50

**O**

Off Grid Track Corners 31
On-The-Fly 19
OPPOSITE 61
ORIGIN 32, 50

**P**

P/G Heuristic Router 106
Pad Stacks 163
PADS 24
PADS-2000 11
PADS-LOGIC 121
PADS-Shell 11
PAN 34
PARAM(ETERS) 25
PARAMS 118

Part Decal 24, 147, 162
Part Decals 5, 161
Part Outlines 23
Part Type (Using PADS-2000 CREATE) 151
PART TYPES (Using PLIB) 148
Partially Routed 110
PARTS LIBRARY 139
PARTS LIST 139
Parts Lists 5
PHO-PLOT 136
PHOTOPLOTTER ARTWORK 135
PLACEMENT 4
PLACEMENT EVALUATORS 78
Placement Matrix 70
Plane 30
Plot Type 132
Pop-up 18
Power 91
Power and Ground Heuristic Router 106
Power and Ground Planes 187
POWER AND GROUND ROUTING 104
Power plane 168
Prompt Line 14

**R**

REAL WIDTH 26, 91
REF-DES 79
Rename Matrix 79
RENAMING OF PARTS 78
Repaint 33
REPORTS 5, 110, 117
Reset 33
Ripup 30
ROTATE 61
ROUTE 5
Routing Grid 31
ROUTING GRID SHIFT 111
RUT CONN 91

**S**

Schema 19
SCREEN MANIPULATION 33
Screen Presentation 9
SEL TEAR 92
Selecting Transformation Options 133
SET ANG 61
SETUP 4, 21, 50, 55, 118
Share 29
Short Memory 108
Shove 30, 91
SHOVE ROUTER 101
SHOW CURRENT LEVEL 25
Signal Heuristic Router 109
SIZE 24
SIZES 24
SMD 183
SMD DESIGN 183
SOLDER MASK 134
Space Check 118
Spacing 5
SPIN 61
SR PARMS 30
STATS 110
SYNC TEST 103
SYSTEM LIBRARY FILES 207

T

2-D Lines 5
T-ROUTING 98, 100
TEAR DROP 26
Tear Drops 118
Text 5, 51
TEXT STRING 51
THERMAL LINE WIDTH 26
Thousandths of an Inch 31
Thru Via 5, 167
TIE PLANE 120
To the Schematic 121
TOP 23
Tracks 5
Tracks on Planes 180

TRAINING BOARD PARTS LIST 42
TRAINING BOARD SCHEMATIC 38
TUTORIAL 1 9
TUTORIAL 2 49
TUTORIAL 3 89
TUTORIAL 4 117
TUTORIAL 5 131
TUTORIAL 6 139
TUTORIAL 7 167
TUTORIAL 8 183
TUTORIAL 9 205
TUTORIAL 10 211

**U**

UNGLUE 63
UNITS 31
UNROUTE 92
User Macros 212

**V**

Vertical 29
VIA MINIMIZATION 110
Via Shift Allowed 31
Virtual Memory 11

**W**

Working Area 13, 14
"Working xx" 20

# APPENDIX 1

# POSSIBLE ERRORS AND CAUSES

# Appendix 1

# POSSIBLE ERRORS AND CAUSES

This Appendix is devoted to identifying possible error conditions, their indications and, where necessary, the correction effort.

## 1. Spacing Errors on Artwork

You have designed the board, post processed the artwork and even built a number of boards, when you find many shorts and spacing conditions. (Yes, indeed, this has happened!)

Possible Causes:

1. You did not run Space Check before plotting Artwork.
2. You ran Space Check but did not have the error symbols turned ON and thus did not realize you had errors.

Corrective Action:

Always run Space Check just before Artwork. Turn ALL Display items OFF except errors. Check for error symbols.

## 2. Incorrect Photoplotter Aperture

The Problem: Upon inspection of artwork, you find that it is different from the Verify plot created in the CAM routine.

The Cause: The photoplotter aperture has not used the same aperture as specified in the CAM Artwork Routine.

The Fix: Be sure to include the Aperture Assignment Table with your Artwork file for use by the photoplotter house.

## 3. Imbedded Tracks in Poured Copper Area

The Problem: The tracks on a Poured Copper area are not separated from the copper.

The Cause: You have not set a "clearance" around items in the Poured Copper area.

The Fix: Set a Copper Pour clearance in SETUP, PARAMETERS.

## 4. Not Enough Memory

The Problem: You get an error message "Not Enough Memory"

The Cause:
1. You have not allocated sufficient Memory for both the board database and the program.

2. Previous crashes have created "clusters" on the hard disk and you have not run CHKDSK/F.

## 5. Routed Tracks Disappear

The Problem: You have routed all the Memory tracks (or other tracks) in AUTO ROUTE, the prompt message indicates they have been routed, yet upon exiting from AUTO ROUTE, the tracks are connections rather than routed tracks.

The Cause: You have not allocated enough memory to route the board and save the database. As a result, the routed tracks are not saved..

The Fix: Allocate more Memory.

## 6. No Cursor Movement

The Problem: After loading the PADS-2000 program, the cursor will not move when the mouse is moved.

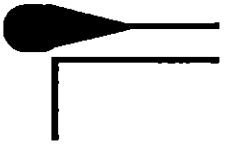
The Cause: You have not selected the correct Mouse Driver.

The Fix: Using PINSTALL, reset the hardware for the correct Mouse Driver.

## 7. Space Errors with Tear Drop Pads

The Problem: After turning on "Tear Drop Pads" and running Space Check, you see some Spacing errors.

The Cause: Tear Drop pads are put in after Routing, therefore, the Router does NOT consider Tear Drop Pads when routing. As a result, in the instance of a Tear Drop Pad, and an adjacent 90° corner, thus:



occasionally there may be a small spacing error of approximately 1.25 mils.

The Fix:
1. Turn off the affected Tear Drop Pad, or
2. Modify the adjacent track corner, or
3. Ignore the small error.

## 8. No Route with Memory Connection

The Problem: You route a board using the ROUTE, AUTOROUTE and SHORT MEMORY, and No Memory connections are selected or routed.

The Cause: SHORT MEMORY is defined as connections whose length is 600 mils or less.

The Fix: Replace the parts such that the length of a Memory connection is not greater than 0.6" and reroute.

## 9. Copper Flood doesn't Completely Flood

The Problem: You are creating a very large area of COPPER POUR, you receive the message:

**UNABLE TO SET DISPLAY LIST**

and the area only partially floods.

The Cause: There is not enough free space on your Hard Disk to contain the board plus the large copper area.

The Fix: You must Free up additional space on the hard disk.

## 10. Not Enough Memory (Disk Message)

The Problem: You receipt prompt message similar to the following:

**CAN'T WRITE DISPLAY FILE**

**I/O ERROR WRITING JOB FILE**

The Cause: You need more RAM memory or more free disk space or both.

The Fix: Add more RAM or free up additional disk space.

## 11. Can't Find Part

The Problem: You read in a Part List and get the message:

**CAN'T FIND PART**

The Cause: Most likely you have used the Part Decal instead of the Part Type info like this:

U1 DIP14
U2 DIP 20

The Fix: Use the Part Type like this:

| | |
|---|---|
| U1 | 74LS04 |
| U2 | 6167 |

# APPENDIX 2
# PADS-2000 Self -Training Manual Supplement

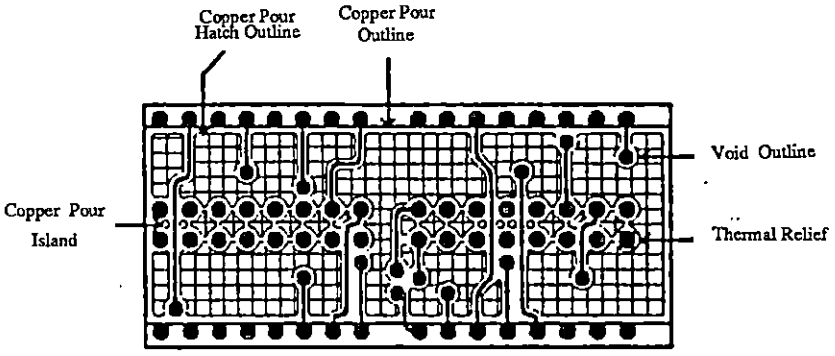
**New Features in Version 3:**

- Improved copper flood with smoothing function and island removal
- Padstack modification for a single part reference instead of all parts with the same decal
- Curved track capability
- Automatic mitring of corners to 45 degree or curved radius
- "Flying" re-connect. During placement, the system will automatically re-connect the ratsnest for shortest distance as the part is moved around.

Here is a tutorial for the new features:

# Copper Flood

Copper Flood incorporates new features and operation. To understand the new capabilities it is necessary to understand some new terminology. Refer to the following diagram:




The diagram shows an area of the board which has been flooded. It contains tracks, pads, a via, a cutout and the copper pour itself. These definitions are used when describing the objects in the diagram:

Copper Pour Outline = The overall shape of the copper flood which was designed before the flood operation.

Hatch Outline = The flooding process builds one or more hatch outlines inside the original pour outline. The hatch outline is "carved" around the tracks and pads inside the pour outline.

Void Outline = A void outline is a clearance that is created inside a hatch outline to disconnect the copper from a pad or track inside it.

Cutout = A predefined void outline which will inhibit copper flood. A cutout can be any user defined shape including arcs and circles.

Pad Thermal Relief = A four pronged connection between a pad and the copper flood. This will be automatically created by the copper flood when the netname of the copper matches the netname of the pad (same as in the past). If a thermal relief extension doesn't intersect a copper pour polygon, then that individual extension will be deleted from the thermal. An error report will be generated describing the thermal relief pads which only have one, or less, extensions on the pad.

Isolated Island = An area of copper which is disconnected from the bulk of the copper pour. These will be created when two components are close to each other and the clearance of the copper pour prevents it from reaching the small areas in between.

## Changes in the SETUP for Copper Pour

The SETUP/PARAMS form has some new options which control the copper pour capability:

Copper Pour Hatch Display: This is either ON or OFF. This setting controls the visibility of the copper pour hatching. With Hatch display OFF, Copper Pour outlines may be edited. With Hatch display ON, Hatch outlines may be edited. It is therefore very important to select the correct display before editing Hatch or Pour outlines.

Copper Hatch Mode: This can be OFF, Normal or See Thru. This option is designed purely to improve the redraw speed and graphics appearance.

Copper Pour Clearance: This is set in the current units. A typical value might be 13 mils and that would be the space allowed as clearance between the copper and the tracks and pads inside it.

Copper Hatch Direction: This is either H/V (horizontal/vertical) or DIAG. This setting determines the pattern of the copper hatching within the flooded area.

Smoothing Radius: This sets the ratio of the outline width and the radius used to trace around obstacles. A larger value will amount to a smoother outline around tracks and pads.

Minimum Hatch Area: This sets the island removal. Typical value is between 100 and 200 mils. Anything smaller than this is removed automatically.

To further illustrate the versitility of the copper pour routines;

1. Load the example COPJOB1.JOB
2. Go to the PARAMS (F2)/SETUP (F4) form and set the Copper Pour Parameters:

   Copper Pour Hatch Display = ON

   Copper Hatch Mode = Normal

   Copper Pour Clearance = 13 Mils

   Copper Hatch Direction = DIAG

   Smoothing Radius = 0.1

   Minimum Hatch Area = 0
3. Go to the CREATE (F3) menu and select COP_POUR (F6).
4. Create a rectangle around the components and tracks (Standard 2-D Line editing).
5. Select FLOOD (F9) from the menu. You now have three choices:

   SELECT (F1) - this is the default and allows you to point at a particular pour outline and flood it. In this way complete control over the flood process is possible.

   FLD_ALL (F2) - As it says, this re-floods every area on the board. BE VERY CAREFUL WITH THIS OPTION as it destroys any local editing that you have done on existing items. This option is mainly used as the initial flood option once all copper areas have been defined.

   FAST_FLD (F3) - This option refloods only those areas which are new (or modified) and therefore need flooding. Any area that is already flooded will be left alone. This option is faster since it does not waste time on existing floods. Also, it preserves specially edited items. This is the preferred option for flooding at a late stage in the design.
6. Select FLD_ALL (F2).

7. Inspect the flooded area and notice how copper has "carved" around the tracks and pads. Notice also that some small islands have been created due to the close placement of the parts on the example.

8. Exit to the CREATE (F3), COP_POUR (F6) menu and use the delete function to remove the islands. This is a new feature and is very useful for cleaning up the copper pour. Simply point the cursor at the offending outline and select DELETE (F7).

9. Also try the MODIFY (F6) function. This is much the same as the standard line editing functions and allows you to move corners and segments of Hatch outlines.

10. After you modify the Hatch outline, notice that the copper hatching inside it disappears. It is necessary to rebuild this hatching.

11. Select HATCH (F8). You will see three options:

SELECT - This is much the same as FLOOD, SELECT but hatching is NOT intelligent. If you have moved the outline over a pad, this will simply be shorted out by the new hatch.

HATCHALL - Just like FLOOD but with unintelligent hatching.

FAST_HAT - Only hatches the modified outlines.

12. In some cases, you may have modified the outline so that it doesn't conflict with tracks and pads. In this case it is possible to re-flood a single HATCH OUTLINE (NOT Pour Outline). To do this, the Copper Hatch Mode and the Copper Pour Display must be turned on. You can then select FLOOD (F9)/ SELECT (F1) and point at the hatch outline. Clearances will then be made around all obstacles.

13. Make several additional to make sure that you are clear about the basic operations of Copper Pour.

## Advanced Copper Pour Features

1. Now suppose that you wanted to attach a pad to the copper pour area. The example has some pads that are connected to GND. Type "N GND <cr>" to see the pads.

2. Following on from the previous example, go to MODIFY (F5), NET_NAME (F9) and try to select the outline of the copper pour. . .

   . . . Chances are that you got the message "Function can only be performed on Pour Outlines". This is because you have Copper Pour Hatch Display = ON. Therefore, Copper Pour outlines are not visible. You need to turn Hatch Display off before adding a netname to the copper pour area.

3. You could use the SETUP (F2), PARAMS (F4) form but a shortcut is the "PO" modeless command. (Pour Outline). This toggles the display of the copper hatching without using the setup form. Try typing "PO <cr>" to see the effect.

4. Once you have turned hatch display off, you can set the name of the copper pour to GND. Do this and reflood using any option you like from the FLOOD (F9) menu.

5. You need to turn the Copper Pour Hatch Display back on again to see the result. Notice that the pads that are connected to GND now have thermal relief marks on top of them which connect them to the plane.

6. You may also try adding a cutout with the CREATE (F3), CUT_OUT (F7) menu. To see the result, you need to reflood the hatch outline (or reflood the whole area).

# Curved Tracks

PADS-2000 Version 3 has the capability of routing a board using curved track and mitred track technology. To understand how curved tracks operate:

1. Load the DEMO job.

2. Enter the ROUTE (F5) menu and pick up a connection with RUT_CONN (F1).

3. Put in a couple of regular corners first and then try ADD_ARC (F5). Notice the following:

At first the arc snaps to a 90 or 180 degree turn (as shown in the coordinate display)

The Angle (F3) function allows any angle of turn to be included.

The coordinate display shows the radius and angle of the arc dynamically.

4. Finish the arc by selecting ADD_CRN (F1) and stretch a new segment away from the arc.

5. Notice how the arc endpoint remains completely fixed.

6. Now type "A <CR>". This invokes the adjust arc mode.

7. Select F3 (angle) and move the segment around. This is a very powerful feature. Notice:

- The arc behaves as if connected to a "crowbar" represented by the new segment and bends accordingly. This allows you to judge the arc roughly and then position the next straight line segment with a smooth continuous connection guaranteed.
- The system may stop temporarily if you move to a point that cannot actually be reached physically by the arc. It will recover as soon as the cursor is moved back to a "legal" position.

8. Finish the track after inserting one or two curves. Note that if you find the animation of the whole track irritating, use ENDNOVIA/ENDVIA (F8) to temporarily drop the track and then immediately pick the track up again. This will stop the "flickering" of the previously routed section of the connection.

## Modifying a Curved Trace

The modifying process is accessed in the normal way through the ROUTE menu. Proceed as follows:

1. Select MODIFY (F5) and MOVE_CRN (F1) while pointing at the arc.

2. You now get a prompt on the command line indicating the actions COMPLETE (F1), RADIUS (F3) and CVT_ARC (F4).

3. Move the corner around and notice how the radius stays constant.

4. Select RADIUS (F3) and observe how the radius is now changed by movement. This is a toggle, so you can easily setup the radius and then position is using the same operation.

5. The CVT_ARC (F4) key toggles between ARC and corner. Press a few times while moving the cursor and notice how you can position the vertex correctly before introducing the correct radius. This reacts something like the AutoCAD filleting capability.

6. Finally, drop the edited curve by pressing COMPLETE (F1).

## Track Mitring

Track mitring is an automatic curving function which curves the vertex between two segments. It is also possible to toggle between 45 degree angled mitres and actual curved ones. To further demonstrate:

1. First check the mitre setup. Select SETUP (F2)\ROUTING (F7)\MITRES (F1) and notice the form that appears.

2. The Mitre Ratio is a numeric value which scales the mitre radius according to the track width. For example a mitre radius of 1.5 will give a 15 mil mitre with a 10 mil track and a 75 mil mitre with a 50 mil track.

3. The minimum radius sets the angle beyond which PADS will not bother to introduce a mitre. It is like saying "if the bend in the trace is smaller than ?? degrees, do not bother to put in a mitre". Of course, 180 degrees between to segments implies a straight line and any angle between the segments will therefore get a mitre. If you set the rule to 150 degrees, this should be a reasonable if arbitrary control.

4. After exiting from the mitre setup form, you can preview the settings at any time by typing "MM <CR>" (mitre menu). This is a toggle function; type MM again to remove.

5. Before routing make sure that all pull downs are displayed since additional information will then be displayed concerning mitring.

6. Now start routing.

7. Once a track is in progress, select a mitre setting by typing "Mn <CR>" where n= 0 to 7. This selects a mitre level. For example M4 will select whatever ratio you setup for level 4.
8. Continue routing and notice that each corner now has a curved mitre.
9. Note that: M0 = mitre off. Also, a track can have multiple mitring levels with no problem.
10. When showing mitring be sure to route a section with ANGLE (F3) selected. This shows how the mitre will move smoothly to connect the two segments at any angle.

## Mitre Editing

Mitre editing is accessed through the MODIFY (F5) menu in ROUTE. If you choose MIT_CRN (F7) you can now change the mitre level of any corner either to the current level or in incremental steps. Therefore to turn a mitre off you would type "M0 <CR>" and then MODIFY (F5), MIT_CRN (F7), MIT_CUR (F1).

It is also possible to use the regular arc modifying routines described above.

Finally, you can toggle the setup between straight bevels and curved mitres by using the SETUP (F2), ROUTING (F7), MITRES (F1) form.

## Flying Re-Connect

To show this feature, simply move a part around the board as you would normally when placing it. In version 3, the "ratsnest" automatically reconnects itself as the part is moved around. Notice the command line which displays the percentage change in connection length from where the part was originally. This lets the designer see how their placement is improving the connectivity of the part. You can turn this function off by selecting the LEN_MIN (F4) key while the part is still at the cursor.

# Shove Router

## Autosteer:

The AUTOSTER (F2) menu has several new features. The SWAP_END (F2), MODE (F5), ENDNOVIA (F6), and ENDVIA (F8) are all new options which can enhance your auto steering capabilities.

*SWAP END (F2) allows you to switch from one end of a net to the other while auto steering.

*MODE (F5). This feature provides you with the ability to choose one of five auto route modes. AUTOPATH ,NOVIAPATH, LATCHED, MANPATH, and GRIDLESS are the five available routing modes.

*ENDNOVIA (F6) allows the user to drop a route without a via at the end of the trace.

ENDVIA (F8) allows the user to drop a route with a via at the end of the route.

*NOTE: These can be accessed through modeless commands also.

## REROUTE:

REROUTE (F6) allows a user to modify an existing trace using the same command options as found in AUTOSTER.

## MODNETS:

The MODNETS (F5) menu has an advanced option which deals with ECL nets.

ECL1 (F6) - When invoked, will follow the same functionality defined in the net attribute section of PADS-2000 (i.e. the loads, source, and terminator will be LENGTH MIN'd in a series or a parallel fashion).

Example:

1) Bring in DEMO.JOB

2) Enter the ROUTE (F5), SHOVE (F4) routine.

3) Select the AUTOSTER (F2) menu and select a connection inside the route box.

4) Next, select the SWAP_END (F2) menu and notice the track is starting from the other end of the two node pair.

5) Select MODE (F5) and notice the available choices.

6) Select GRIDLESS (F5) and notice the style of routing has changed to a 1 mil grid.

7) Exit back to the CRN/FIN (F1) menu and select ENDNOVIA (F6). Notice the route stops with only a connection at the end.

8) If you had selected ENDVIA (F8), the track would have terminated with a via pad and a connection.

9) Place the cursor over one of the routed tracks and select REROUTE (F6). Notice the segment highlights and you are able to redefine the track as needed.

10) Finally, place the cursor over U4.13. Go into the STRATEGY (F9), MODNETS (F5) and select the net 11. Next select ECL (F6) and answer "YES" to the prompt. This will now control how the net will be routed to ensure the source loads and terminators remain as they are.